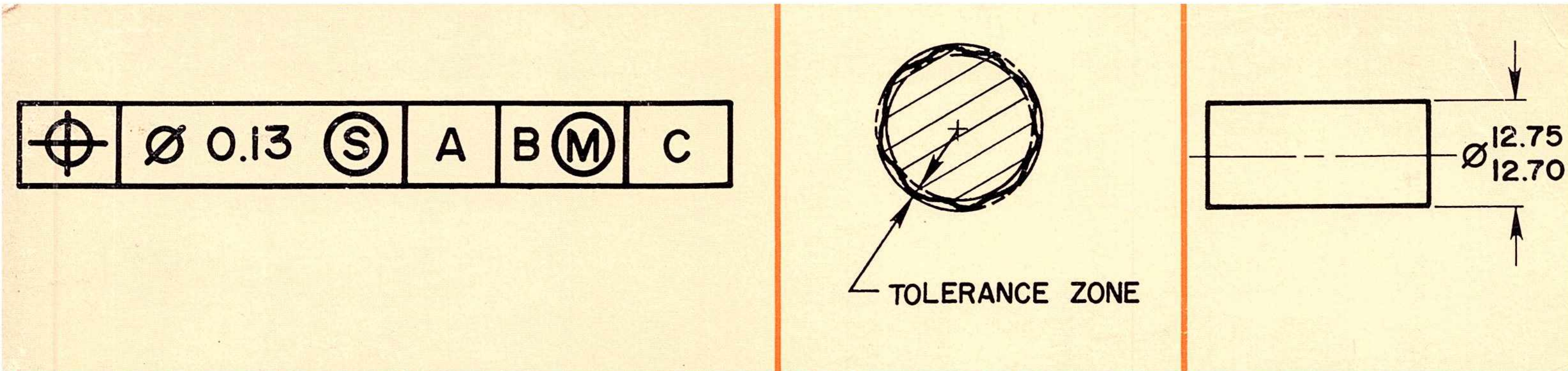
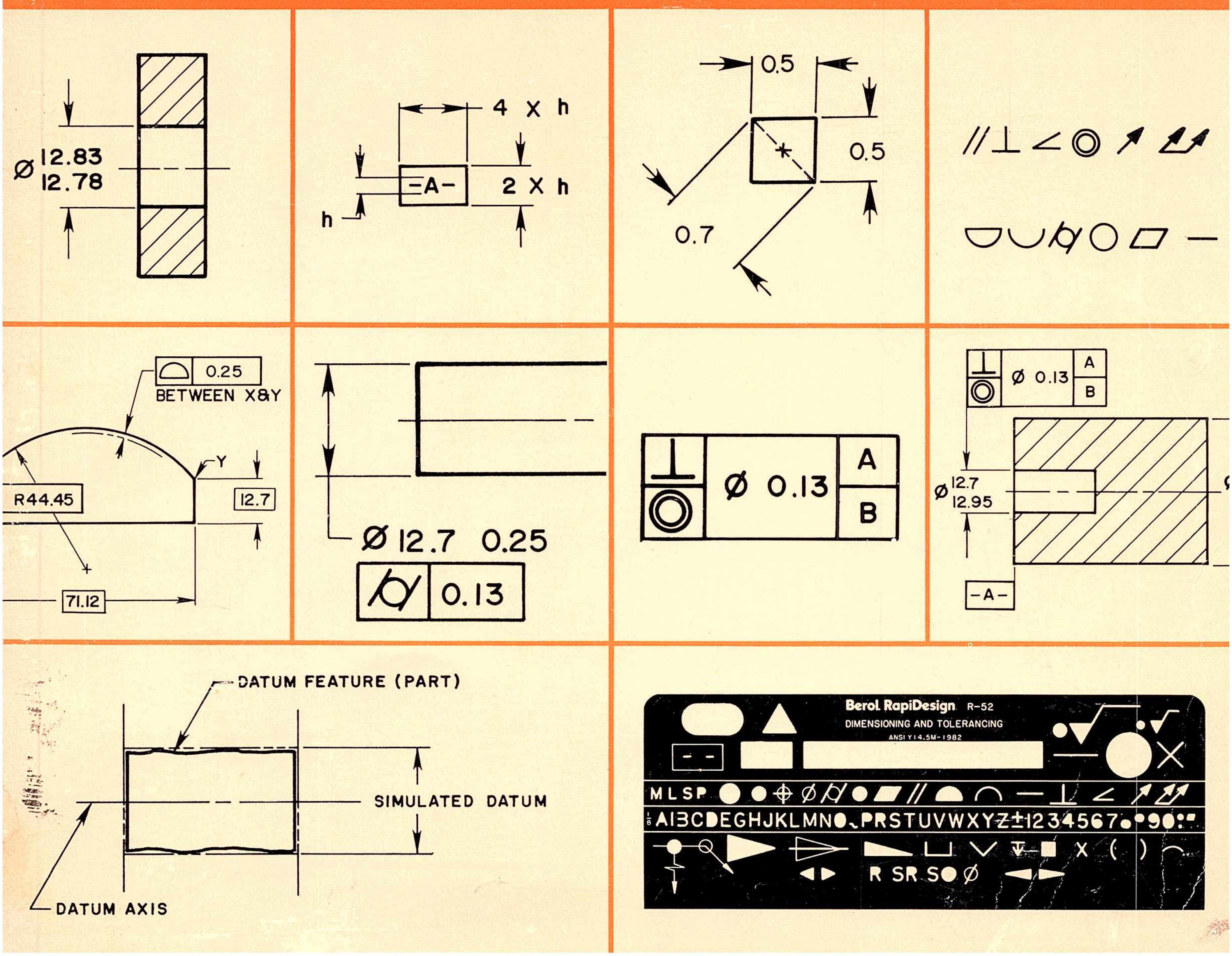


Geometric Dimensioning and Tolerancing

DAVID A. MADSEN



Geometric Dimensioning and Tolerancing

basic fundamentals

by
David A. Madsen
Clackamas Community College
Oregon City, Oregon

South Holland, Illinois
THE GOODHEART-WILLCOX COMPANY, INC.
Publishers

INTRODUCTION

This write-in text is meant to provide you with the basics of geometric dimensioning and tolerancing. The goal of this workbook is to provide you with a logical sequence of learning activities that will lead you from conventional dimensioning and tolerancing into geometric dimensioning and tolerancing on a fundamental level.

The material presented will provide a beginning to a more in-depth study of methodology as the drafter becomes involved in extensive use of the techniques. If you plan a more in-depth study of geometric dimensioning and tolerancing, you should purchase a copy of ANSI Y14.5M 1982. Keep this workbook and ANSI Y14.5 by your desk as a quick GDT reference.

Concepts used in this workbook have been established as standards in the documents presented by the American National Standards Institute, ANSI Y14.5M 1982 (published by The American Society of Mechanical Engineers).

You are encouraged to do all problems and post tests using proper drafting techniques. All symbolism should be drawn to scale as demonstrated by the examples. A geometric dimensioning template is helpful. UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE MILLIMETERS.

David A. Madsen

 Library of Congress Catalog Card Number 83–27451. International Standard Book Number 0–87006–461–4.

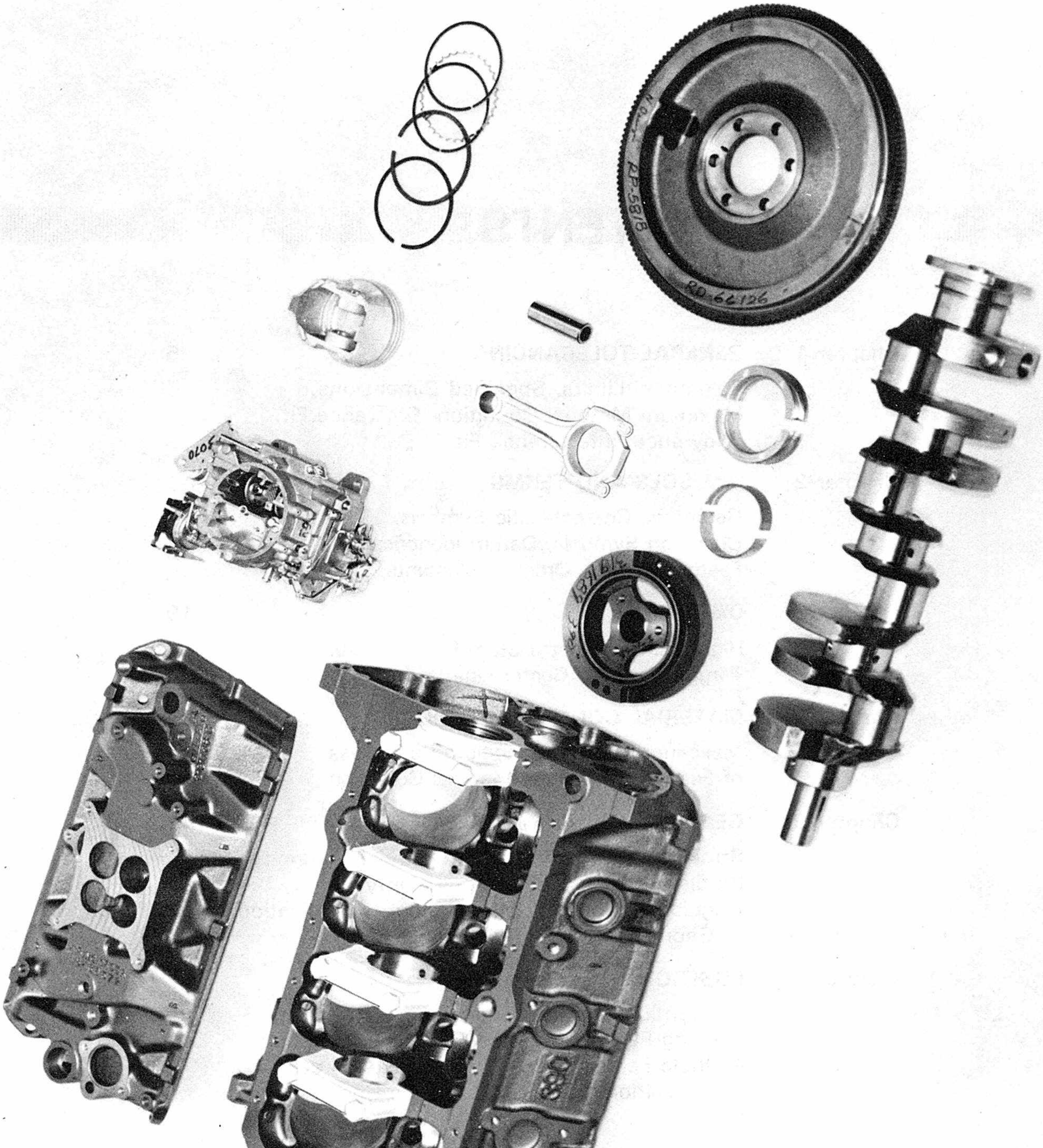
123456789–84–987654

Library of Congress Cataloging in Publication Data

Madsen, David A.
Geometric dimensioning and tolerancing.

1. Engineering drawings—Dimensioning.
2. Tolerance (Engineering) I. Title.
T357.M22 1984 604.2'43 83–27451
ISBN 0–87006–461–4

CONTENTS



Examples of automotive components manufactured from plans using geometric dimensioning and tolerancing.

Chapter 1

GENERAL TOLERANCING

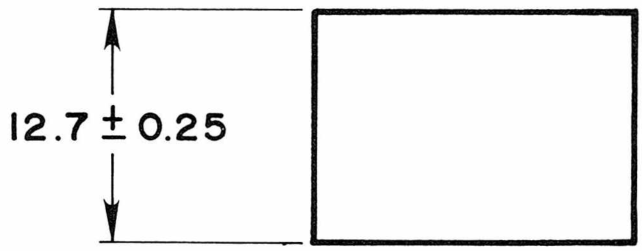
This chapter will cover tolerancing concepts in general. These conventional methods of dimensioning will provide the necessary basic background to begin a study of geometric dimensioning. In future chapters you will see conventional tolerancing and geometric tolerances together.

When mass production methods began, interchangeability was important, but many times parts had to be ''hand selected for fitting.'' Today, industry has faced the reality that in a technological environment, there is no time to do unnecessary individual fitting of parts. Geometric dimensioning, when properly used, will help insure interchangeability of parts. Also, the function of an object dictates the use of geometric dimensioning. Geometric dimensioning will probably not take the place of conventional tolerancing. More important, it will be used as an aid to manufacturing, when parts, due to their function, require its use.

Keep this in mind as you work with the basics of geometric dimensioning: A drafting technician will need to know how to properly represent a geometric dimension. He or she must know how to interpret geometric dimensioning. However, the drafter should not be expected to determine what callout to use and what tolerance to provide in a certain situation; this is the responsibility of a designer or engineer. By the time the drafting technician carries the title designer, he or she will probably have had enough experience and advanced education to provide a proper callout in a given situation.

Realize that the conventional tolerance will usually affect the size of a feature, while the geometric dimension will affect the related geometric characteristic. It is necessary to completely understand general tolerancing before you can be expected to work with geometric dimensioning.

GENERAL TOLERANCING CONSIDERATIONS



Example 1-1

A tolerance is the total amount of size or locational variation allowed in a dimension.

In Example 1-1 the tolerance is ± 0.25 or a total variation of 0.50.
Because, the $\pm$ means + or −, that is 0.25 greater than 12.7 and 0.25 less than 12.7.

The limits of a tolerance dimension are the largest and smallest that the part can be.

From Example 1-1 the limits are $12.7 + 0.25 = 12.95$ and $12.7 - 0.25 = 12.45$.

The specified dimension, also known as nominal size, is the size from which you calculate the limits. The specified dimension from Example 1-1 is 12.7.

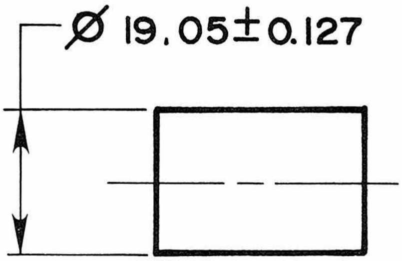
In drafting you might see the tolerance dimension lettered like this

$$12.7 \pm 0.25 \qquad \text{or} \qquad \begin{array}{c}12.95\\12.45\end{array}$$

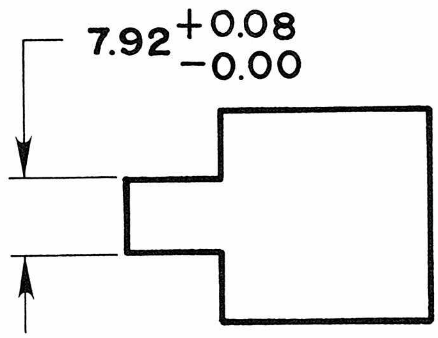
This is a bilateral tolerance 12.7 ± 0.25: it increases and decreases from the specified dimension.

This is a unilateral tolerance $12.7\ {}^{+\,0.25}_{-\ \ .00}$: it increases (or decreases) only in one direction in relationship to the specified dimension.

QUESTIONS:



Part 1



Part 2

(A) What is the tolerance of the dimensions?

Part 1 Part 2

(B) What are the limits?

Part 1 Part 2

(C) Is this a bilateral or unilateral dimension?

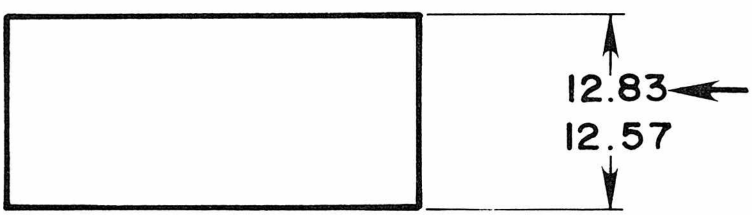
Part 1 Part 2

MAXIMUM MATERIAL CONDITION

Maximum Material Condition is the size of a part or feature that exists when the feature has the most material.

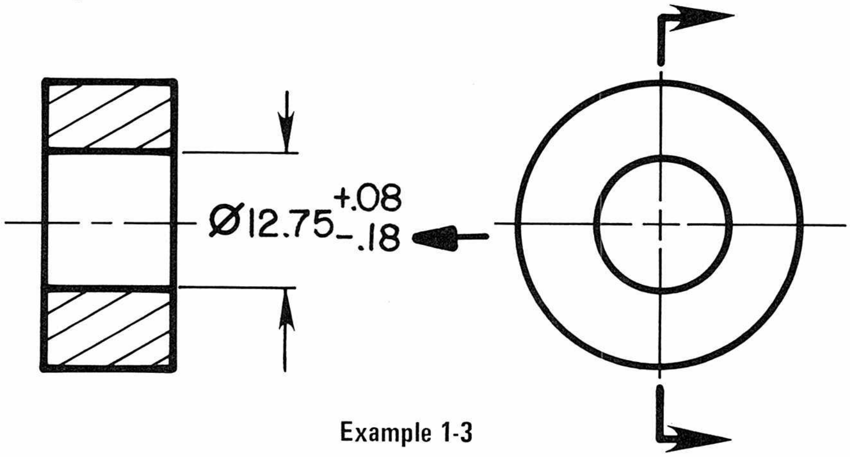
The key words are most material.

An external feature is at Maximum Material Condition at its largest size as illustrated in Example 1-2.



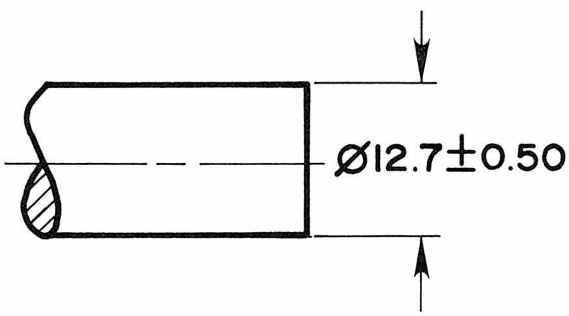
Example 1-2

An internal feature is at Maximum Material Condition at its smallest size as shown in Example 1-3.

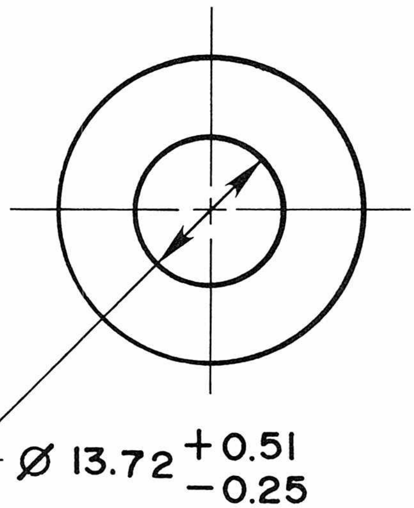


Example 1-3

QUESTIONS:



Part 1



Part 2

1. What is the maximum material condition of Part 1? Part 2?

   Part 1 ____________________.

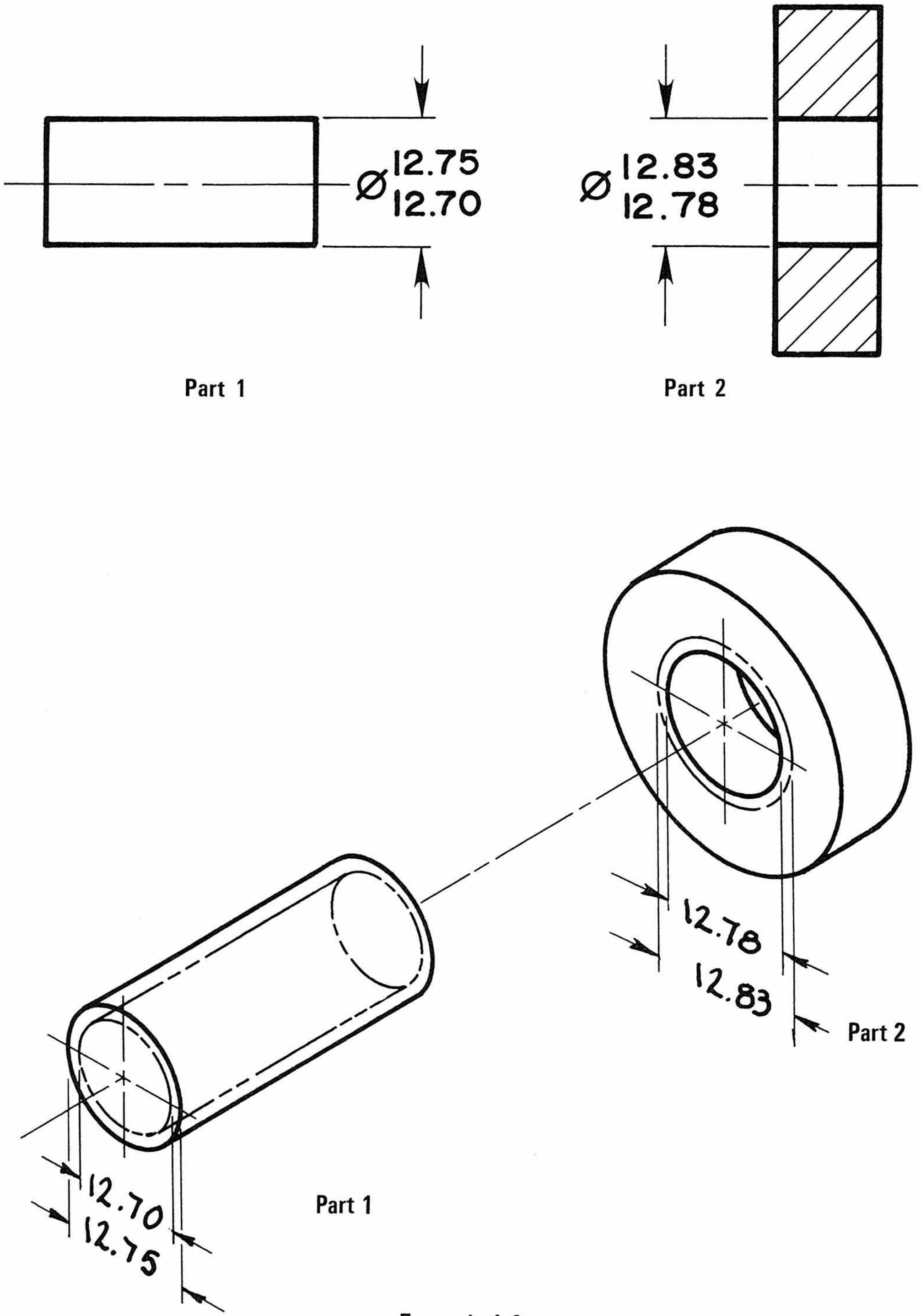
   Part 2 ____________________.

BASIC FITS OF MATING PARTS

For the basic fit of mating parts, the three areas to be studied are: Clearance Fit, Allowance, and Interference Fit.

CLEARANCE FIT

In Example 1-4, Part 1 will fit into Part 2 no matter what size the parts are made within the given tolerance zone. This is a clearance fit.



Example 1-4

ALLOWANCE

Now, if you consider the tightest possible fit between Part 1 and Part 2 in Example 1-4, you will have the allowance or minimum clearance.

Use this formula:

MAX Material Condition (MMC) Hole
− MAX Material Condition Shaft
Allowance

Let's do it:

MMC Hole Part 2	=	12.78
− MMC Shaft Part 1	=	12.75
Allowance	=	0.03

MAXIMUM CLEARANCE

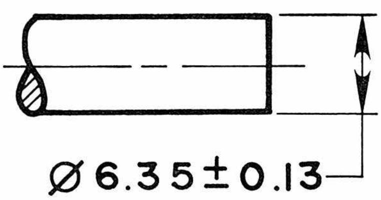
The loosest fit will be the MAXIMUM CLEARANCE and is calculated this way:

Least Material Condition (LMC) Hole
− Least Material Condition Shaft
Maximum Clearance

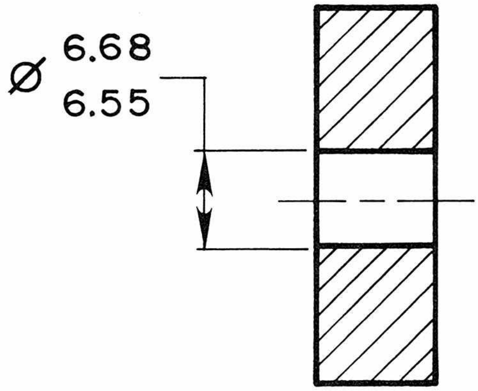
So,

LMC Hole Part 2	=	12.83
− LMC Shaft Part 1	=	12.70
Maximum Clearance	=	0.13

You try it!



Part 3



Part 4

(A) What type of fit is this? ______________________

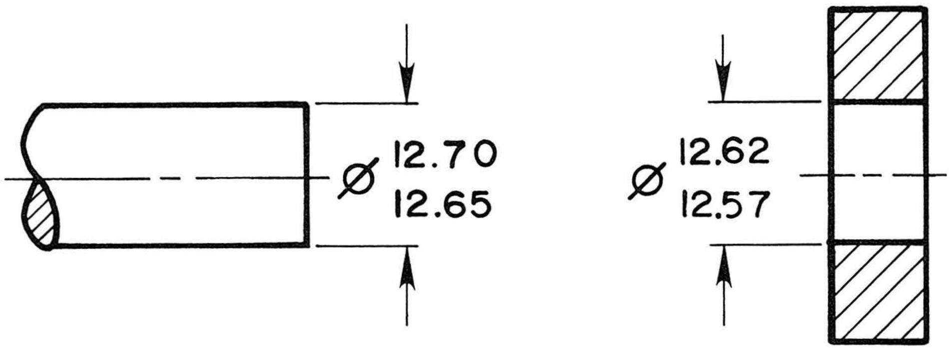
(B) What is the allowance between Part 3 and Part 4? (Show your work.)

(C) What is the maximum clearance between Parts 3 and 4? (Show your work.)

INTERFERENCE FIT

An interference fit is when two parts must be pressed together because the shaft is actually larger than the hole.

Look at the interference fit in Example 1-5.



Example 1-5

At any produced size, the shaft will be larger than the hole.

The smallest amount of interference is:

$$\begin{array}{lr} \text{LMC Shaft} = & 12.65 \\ -\text{LMC Hole} = & \underline{12.62} \\ & 0.03 \end{array}$$

The tightest amount of interference is:

$$\begin{array}{lr} \text{MMC Shaft} = & 12.70 \\ -\text{MMC Hole} = & \underline{12.57} \\ & 0.13 \end{array}$$

GEOMETRIC DIMENSIONING AND TOLERANCING

Post Test 1 Name: ______________________________

1. What is bilateral tolerance? (Show an example.)

2. What is a unilateral tolerance? (Show an example.)

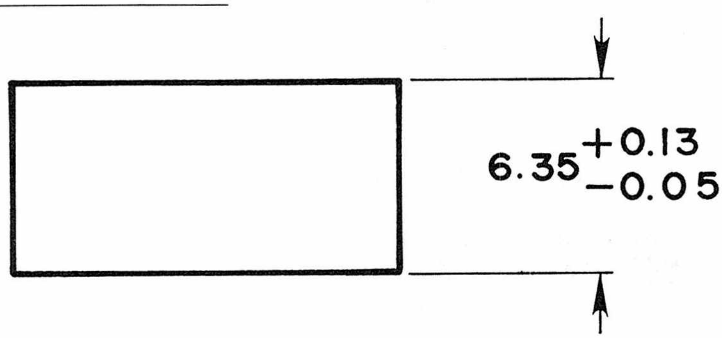
3. What are the limits of this dimension?

$19.050^{+\,0.050}_{-\,0.025}$ ____

____

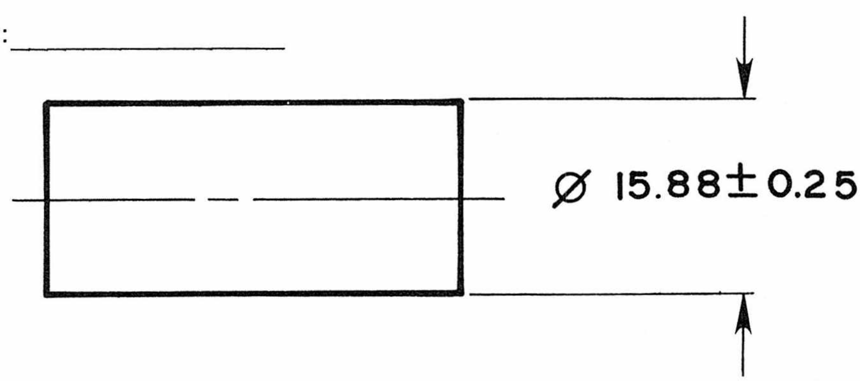
4. What is the specified dimension of this object?

Answer: ____________________



5. What is the Maximum Material Condition of the diameter of this object?

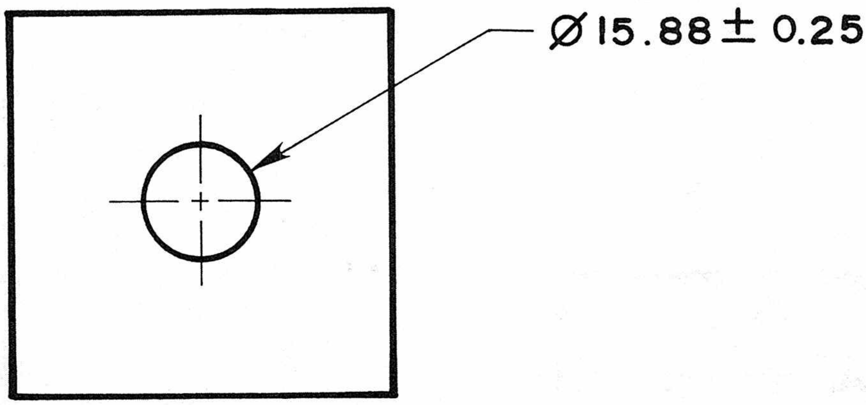
Answer: ____________________



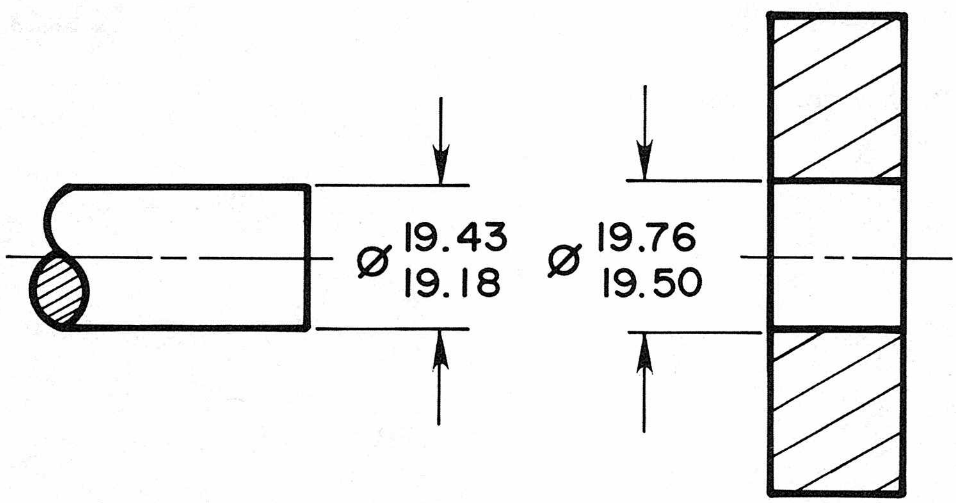
Post Test 1

6. What is the Maximum Material Condition of the hole through this collar?

Answer:____________________



7. What is the allowance between the two parts below? (Show your calculations.)



Answer:____________________

Calculations:

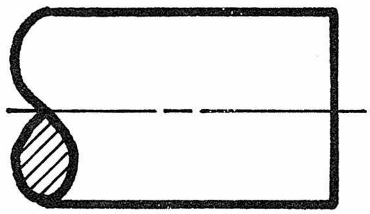
Post Test 1

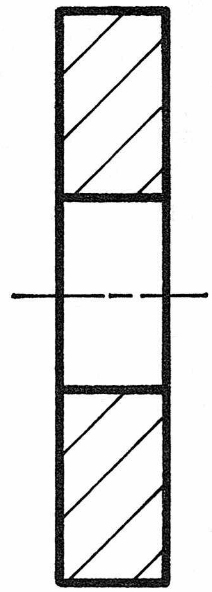
8. Below you will see a shaft and a collar. The specified dimension of the shaft is 12.7. Dimension completely as follows:

   (A) Apply a tolerance of 0.1 (±0.05) to the shaft and 0.254 tolerance to the collar.

   (B) Provide an allowance of 0.05.

   (C) Show dimensions with limits indicated.





   (D) Show your calculations below.

Chapter 2

SYMBOLS AND TERMS

This chapter will aid you in the ability to identify symbols and terms. The main objective here is to recognize the various types of symbols with identification of name, shape, and size. DO NOT concern yourself, at this time, with the meaning of the symbols and terms. This will be covered in later chapters. The symbols that are illustrated are drawn at recommended sizes based on .125 inch high lettering on the drawing. When you draw symbols in the future you should take this into consideration so that your drawings are clear, legible, and represent the proper meaning. Often times, symbols must be supplemented by notes, as you will observe later. When this occurs, every effort should be made to make the total callout as clear as possible.

On all problems in the post tests, draw symbols true to size and shape, using proper drafting practices. A geometric dimensioning and tolerancing template would be helpful.

GEOMETRIC CHARACTERISTIC SYMBOLS

Form
- — Straightness
- ⏥ Flatness
- ○ Circularity
- ⌭ Cylindricity

Profile
- ⌒ Profile of a Line
- ⌓ Profile of a Surface

Orientation
- // Parallelism
- ⊥ Perpendicularity
- ∠ Angularity

Runout
- ↗ or ↗ Circular Runout
- ⌰ or ⌰ Total Runout

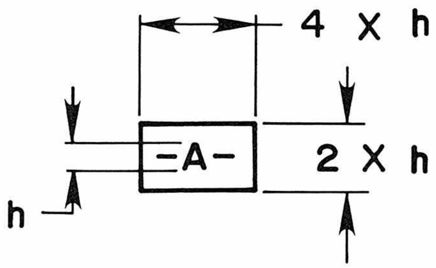
Location
- ⌖ Position
- ◎ Concentricity

Note: All of the symbols are drawn to the actual size and shape recommended by American National Standards Institute (ANSI) based on .125 inch high lettering on your drawing.

MATERIAL CONDITION SYMBOLS

- Ⓜ MMC, Maximum Material Condition
- Ⓢ RFS, Regardless of Feature Size
- Ⓛ LMC, Least Material Condition

DATUM IDENTIFICATION



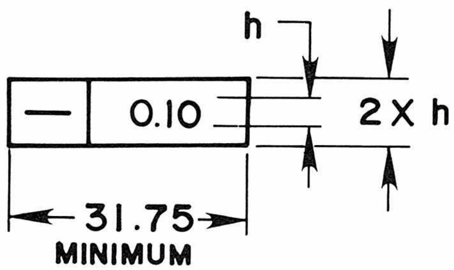
Recommended size (not official)

h = height of lettering

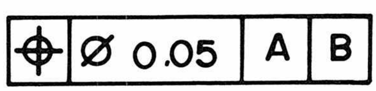
SPECIAL SYMBOLS

- Ⓟ Projected Tolerance Zone
- ∅ Diameter

FEATURE CONTROL FRAME

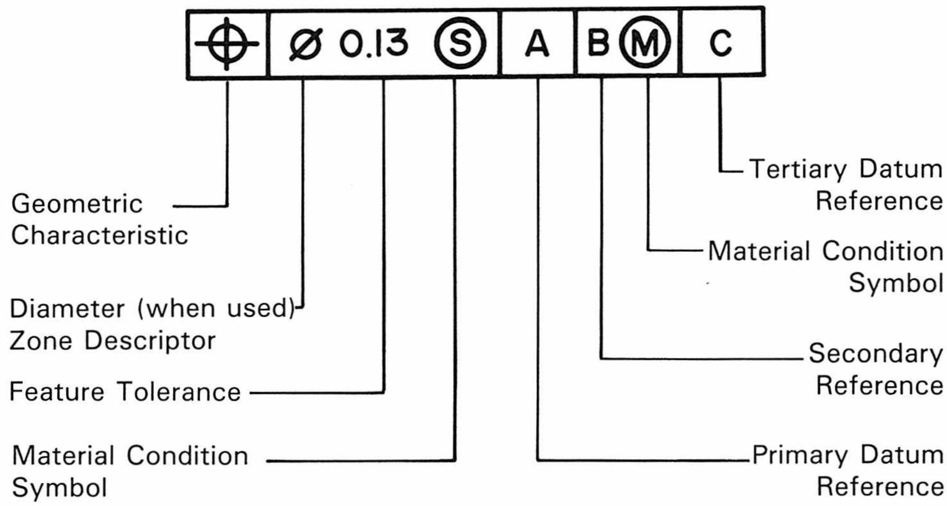


Recommended size (not official)

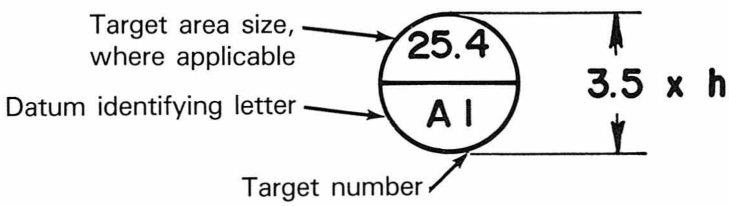


h = height of lettering

ORDER OF ELEMENTS IN A FEATURE CONTROL FRAME



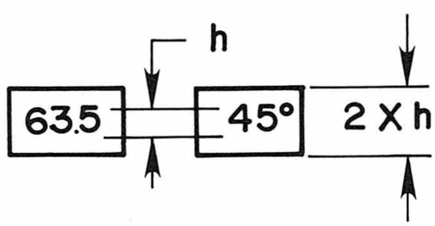
DATUM TARGET SYMBOLS



BASIC

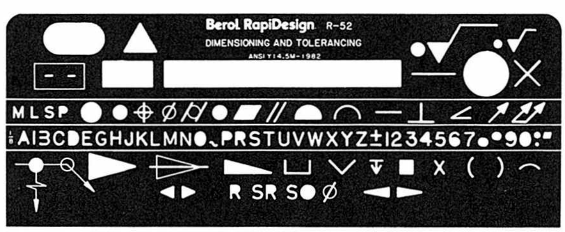
A Basic Dimension is considered a theoretically perfect dimension. Basic Dimensions have no tolerance and are used to describe the theoretically exact size, profile, orientation or location of a feature or datum target. Generally the tolerance for a BASIC dimension is found in a feature control frame; the tolerance is not found in the title block. BASIC dimensions are the basis from which variations are established by tolerances on other dimensions.

Shown as this



DIMENSIONING AND TOLERANCING TEMPLATES

Dimensioning and tolerancing templates are available to help you save time when doing drawings that contain geometric dimensioning symbols. If available, you should use one of these templates to do the post tests in this workbook.



BEROL® RAPIDESIGN®

Example 2-1

GEOMETRIC DIMENSIONING AND TOLERANCING

Post Test 2 Name: ____________________

1. What are the proper symbols for the following form characteristics? (Draw all symbols true size, do not sketch.)

   Flatness:

   Straightness:

   Perpendicularity:

   Parallelism:

   Circularity:

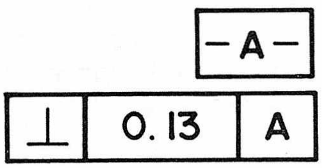
   Position:

2. What is the symbol for MMC?

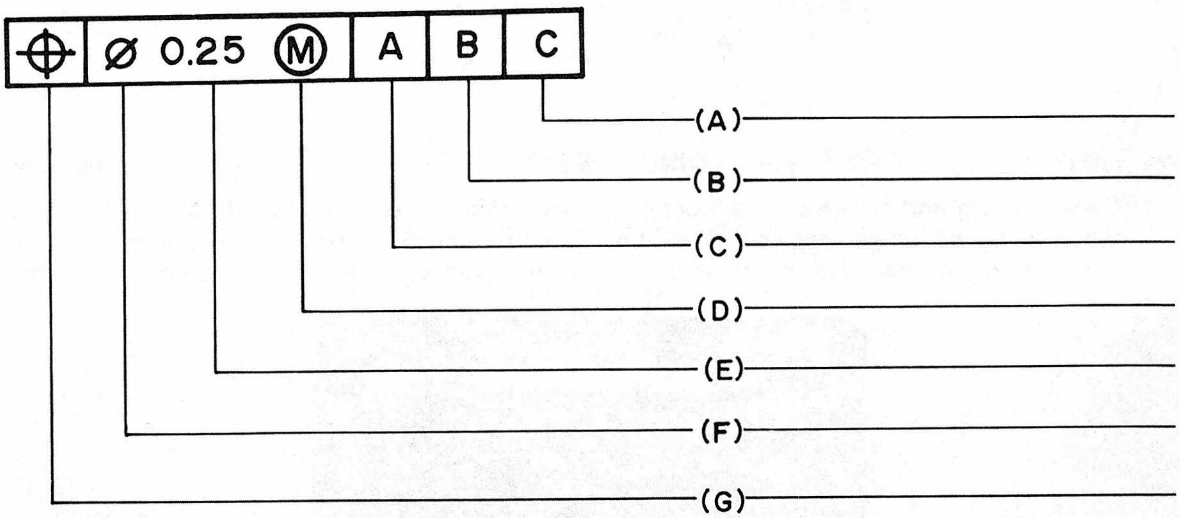
   What is the symbol for RFS?

3. How is a basic dimension shown on a drawing? An example may be used. (Draw true size.)

4. What are these symbols called?



5. Label the parts of this symbol.



Chapter 3

DATUMS

Datums are used as guides for referencing features of an object. You will see that datums are considered to be theoretically perfect planes from which measurements are made. These datums are established by using the finest possible equipment that is available. Examples may be machine tables, surface tables, or specially designed rotation devices. The feature on the object, that is called out as being a datum surface, may not be as perfect as the device that has been selected as the datum plane. For this reason all measurements that are made from datum planes will not take this difference into consideration.

DATUMS

A datum is considered to be a plane, surface point(s), line or axis of an object (called datum features). A datum is assumed to be exact. From this, location dimensions are established. There are many concepts to keep in mind when datums are established. The function of the part, manufacturing processes, methods of inspection, and even the shape of the object are important to think about in establishing datums.

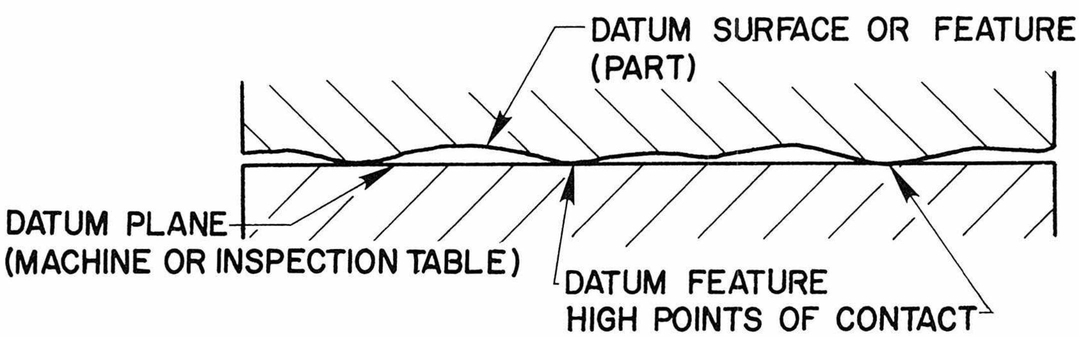
Let's look at some basic datum concepts.

DATUM PLANE/DATUM SURFACE FEATURE (HIGH POINT CONTACT)

A datum plane is a theoretically exact plane.

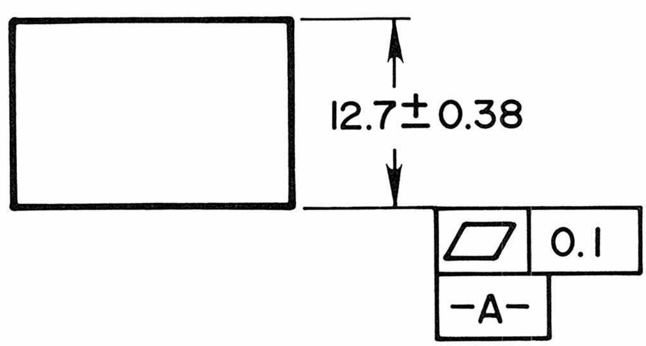
A datum surface is the actual surface of an object that is used to establish a datum plane. Also known as a datum feature.

Look at Example 3-1 and you will see that the datum surface touches the datum plane at the high points of the surface.



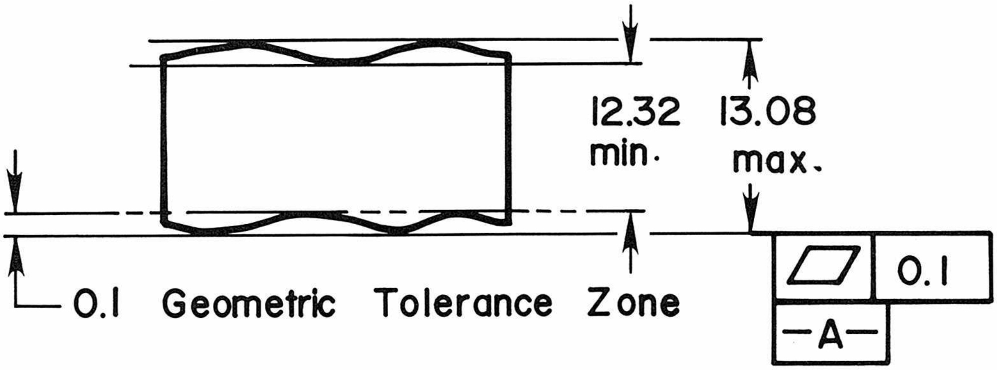
Example 3-1

When high point contact is used to establish a datum, the datum surface may be controlled by flatness. See Example 3-2.



Example 3-2

Look at Example 3-3 and you will see what is really meant by Example 3-2.



Example 3-3

The maximum size that the part can be is the upper limit of the dimensional tolerance.
12.7 + 0.38 = 13.08

The geometric tolerance must be within the size tolerance as shown in Example 3-3.

The minimum size that the part can be is the lower limit of the dimensional tolerance.
12.7 − 0.38 = 12.32

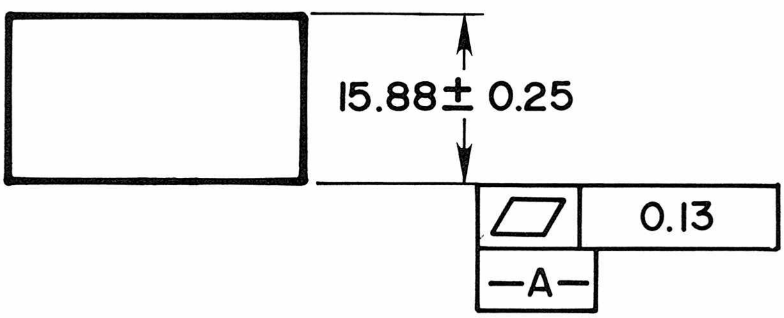
Try this:

From the drawing of the object below, determine the maximum and minimum dimensions that this part could be produced at:

Maximum__________________

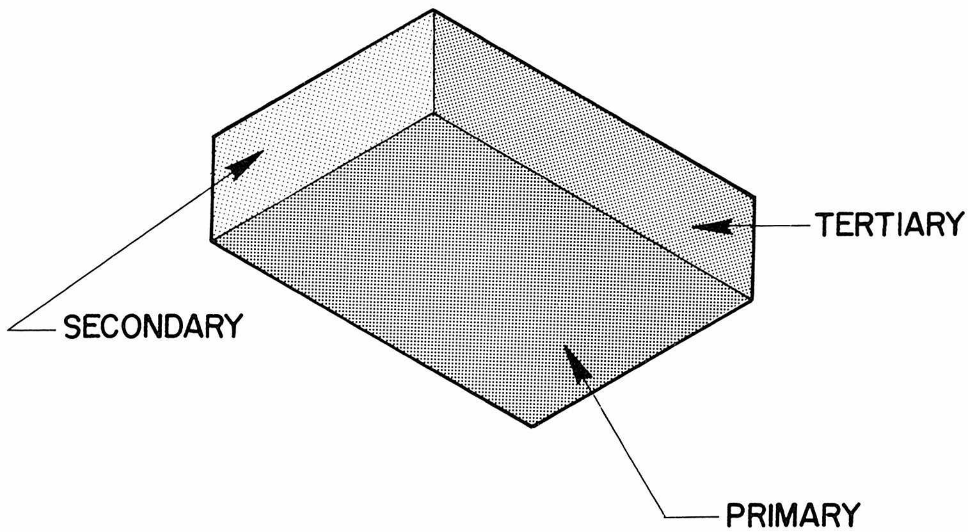
Minimum__________________

Show your work:



DATUM FRAME CONCEPT

A complete datum frame is made up of three datum planes. The first is called the primary, the second is called the secondary, the third is called the tertiary. See Example 3-4.



Example 3-4

The datum planes are established by the datum surfaces of an object as follows:

> The primary datum plane must be established by at least three points of the primary datum surface.
>
> The secondary datum plane must be located by at least two points on the related secondary datum surface.
>
> The tertiary datum plane must be located by at least one point on the related tertiary datum surface.

Selection of the primary, secondary, and tertiary datums is made by the functional importance on the object.

The secondary datum is established with perpendicularity to the primary datum.

The tertiary datum is established with perpendicularity to the primary and secondary datums.

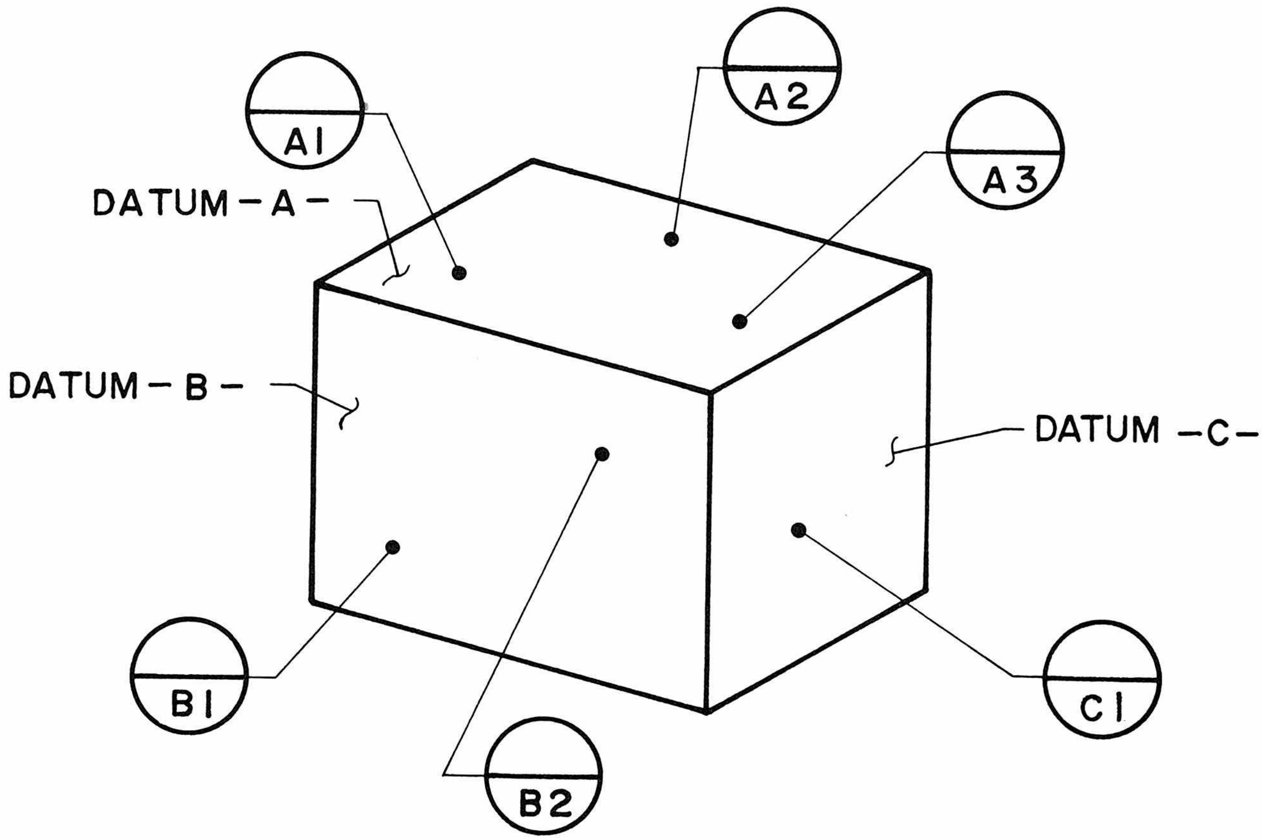
Often times parts may not always have surfaces 90° to each other, such as castings, forgings, or sheet metal. Surfaces with draft or other characteristics may specify an angle other than 90°, for example 87°.

DATUM TARGET SYMBOLS

In a previous discussion you saw that the high points of a datum surface feature were used to establish the related datum plane. This is great when an entire surface is machined to produce this datum. However, often times on a casting, forging, or a large surface, it is not economically possible to machine the entire surface. For this reason, datum targets are established on the drawing using basic or ± dimensions. These datum targets are established by fixtures with pins. These pins contact the part where the datum targets are specified.

> Three points for the primary, two for the secondary, and one for the tertiary datum.

Here is what is meant:



Example 3-5

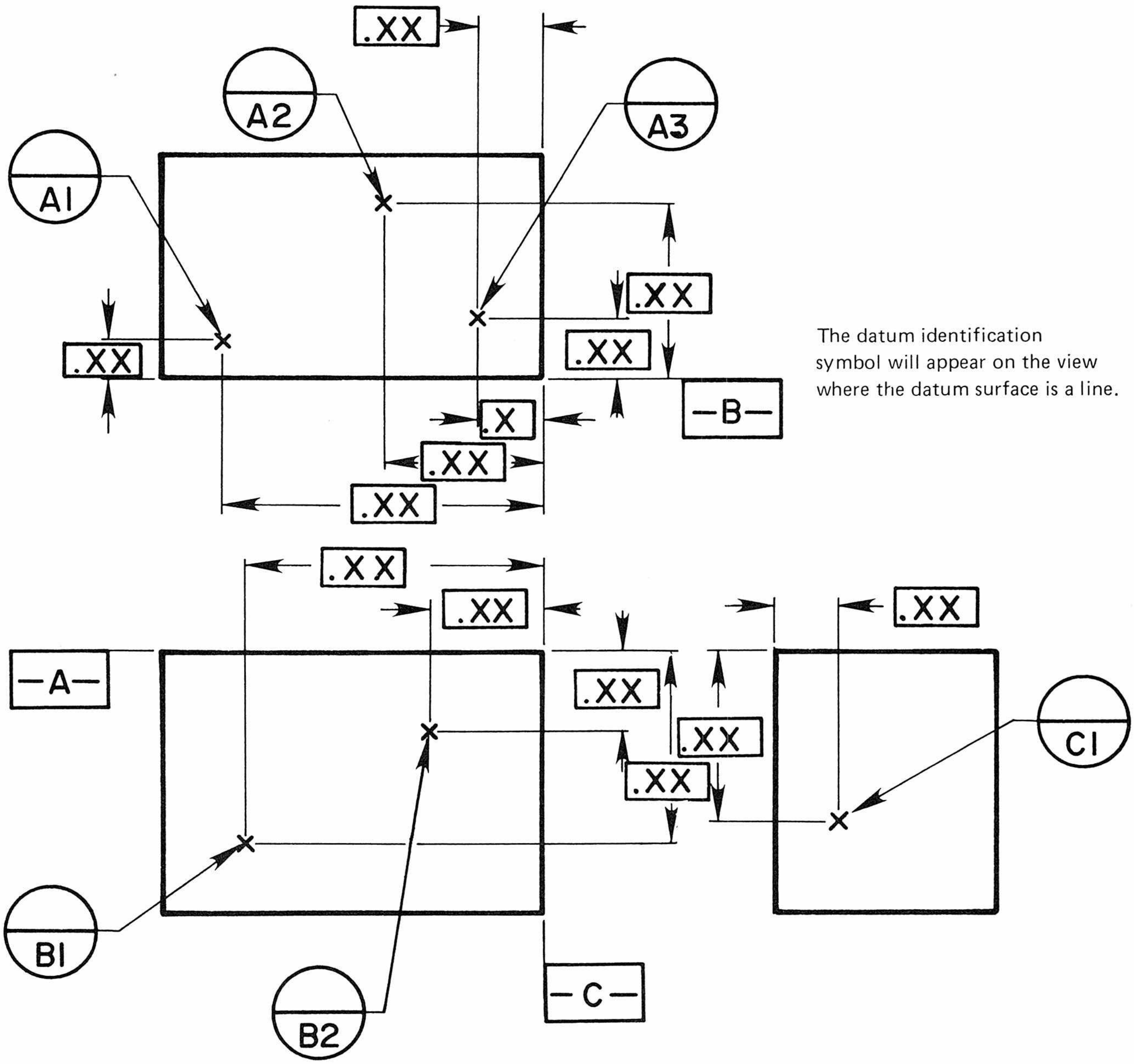
Note that each point is identified with a datum target symbol A3

In the datum target symbol the A identifies the datum.

The 3 identifies the point.

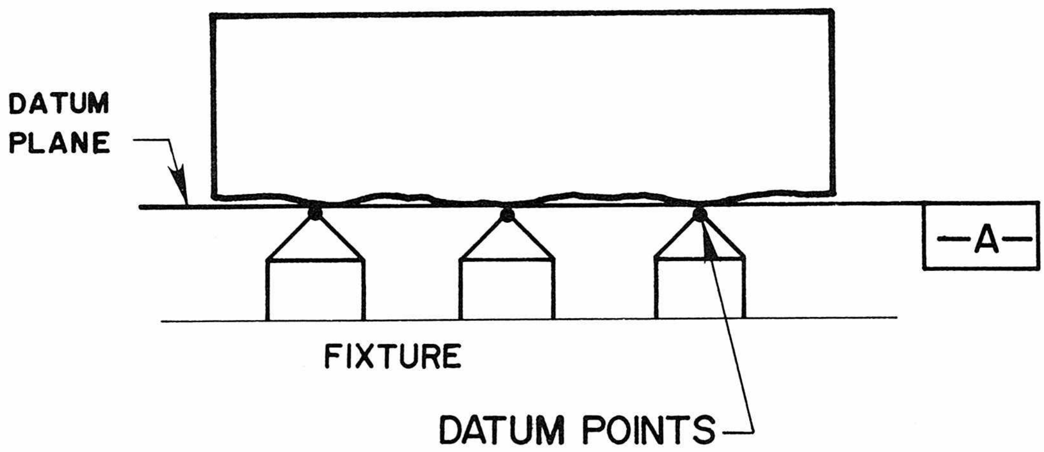
The datum points may be located with BASIC dimensions or ± dimensions.

Let's look at Example 3-6 for a multiview representation of Example 3-5, using BASIC dimensions.



Example 3-6

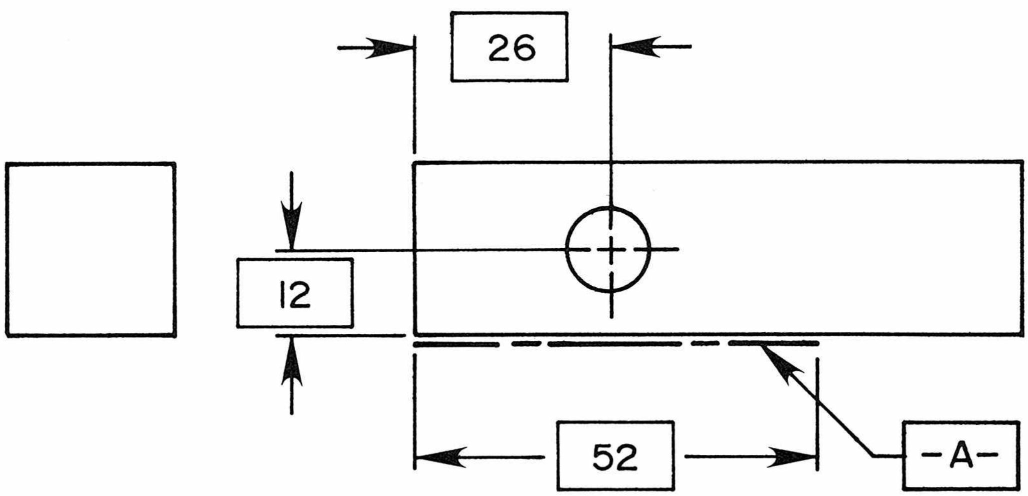
Examine Example 3-7 and you will see that the datum plane is established by the surface at the datum points.



Example 3-7

PARTIAL DATUM SURFACE

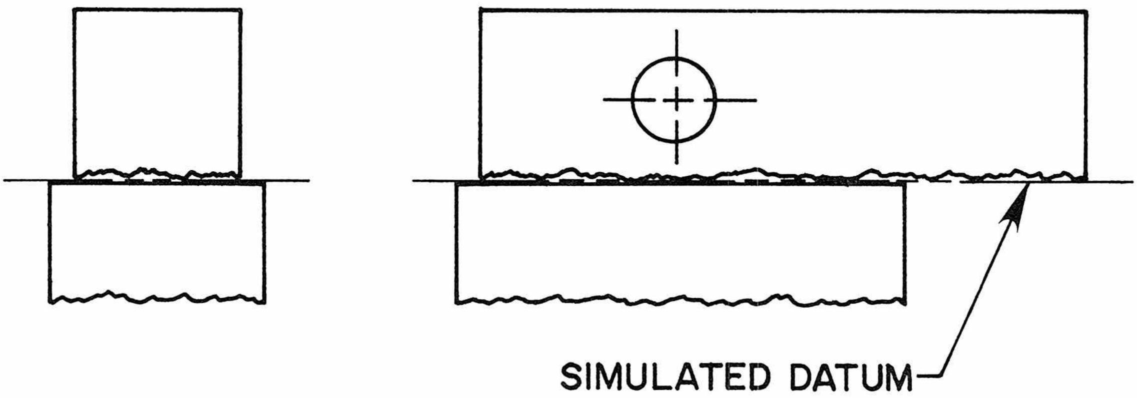
A portion of a surface may be used as a datum rather than the entire surface. This may be accomplished on a drawing as demonstrated in Example 3-8. The chain line denotes the extent of the datum and carries a basic dimension.



Example 3-8

Look at Example 3-9 to see what the partial datum means.

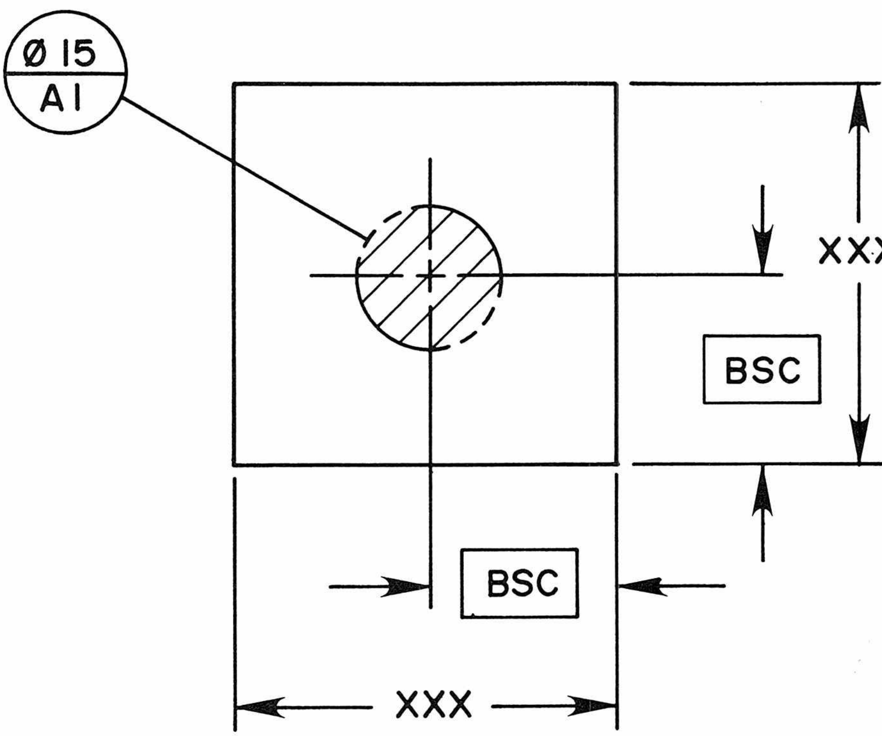
In this example, a datum over the entire length of the part is not necessary for location of the hole.



Example 3-9

DATUM TARGET AREAS

Areas of contact may be used to establish datums. When this is necessary, the shape of the datum target area is created by phantom lines with section lines through the area. The area, if circular, is dimensioned with basic dimensions to the center and the diameter of the area is provided in the upper half of the datum target symbol. Look at Example 3-10 and you will see a representative drawing.



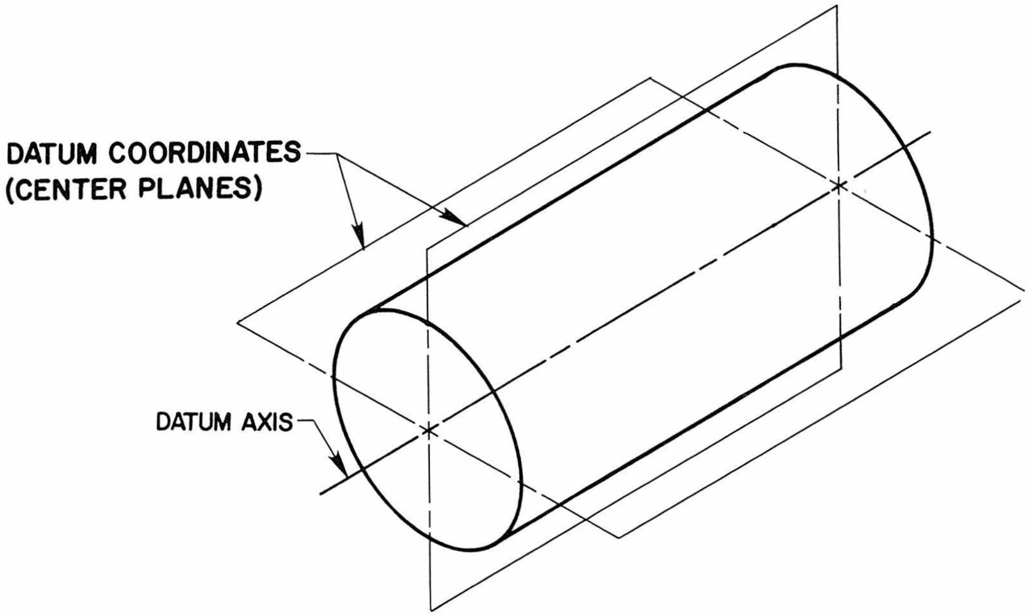
Example 3-10

When the area is too small to draw, then a datum target point is used at the center of the location and the datum target symbol identifies the diameter.

CENTER LINE OR AXIS DATUMS

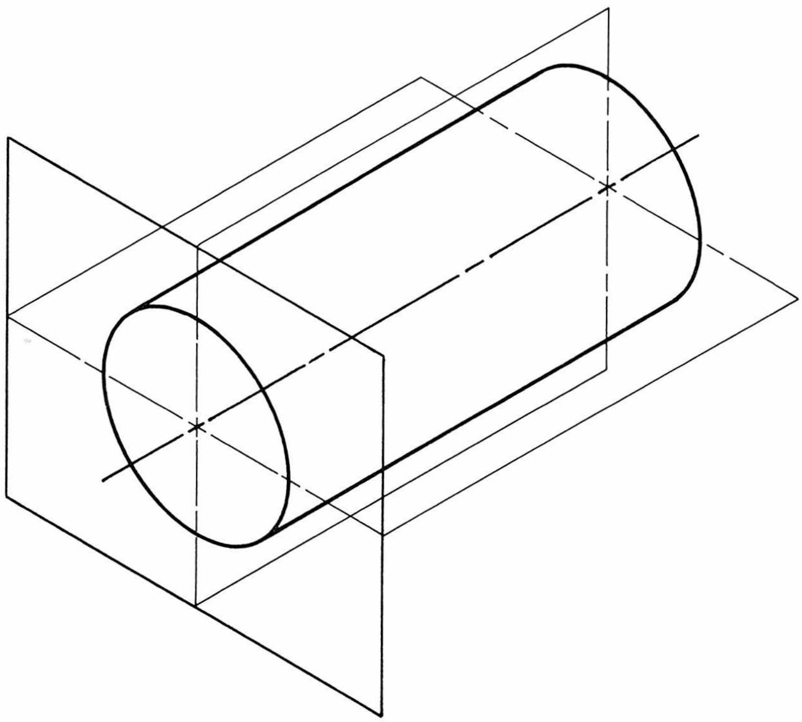
The cylindrical object can be a datum feature. Its center line is known as a datum axis.

Two datum coordinates established by the center of the object are used to create the datum axis. Now examine Example 3-11 and you will see how the two center planes form the datum axis.



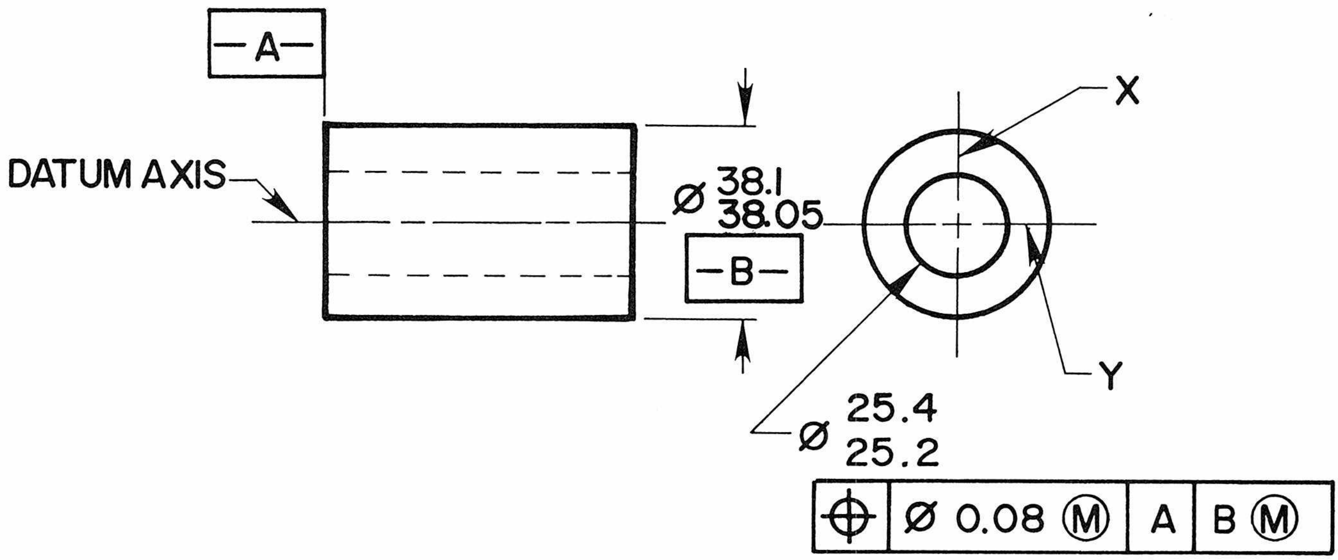
Example 3-11

A third datum plane can be added to the end of the object to establish the datum frame. See Example 3-12.



Example 3-12

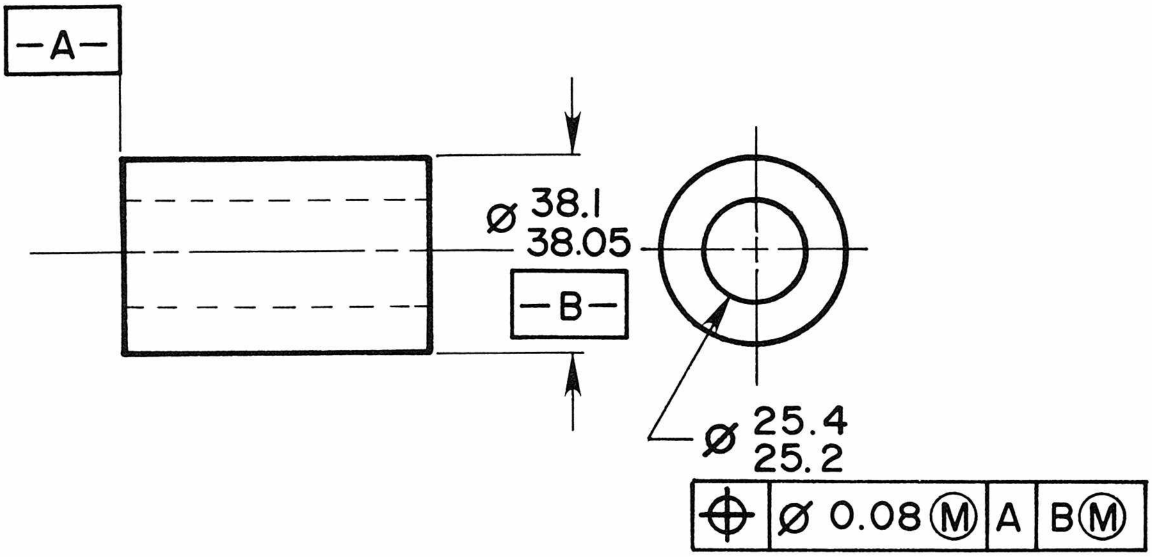
In Example 3-13, you will see how the datum axis can be represented on a drawing.



Example 3-13

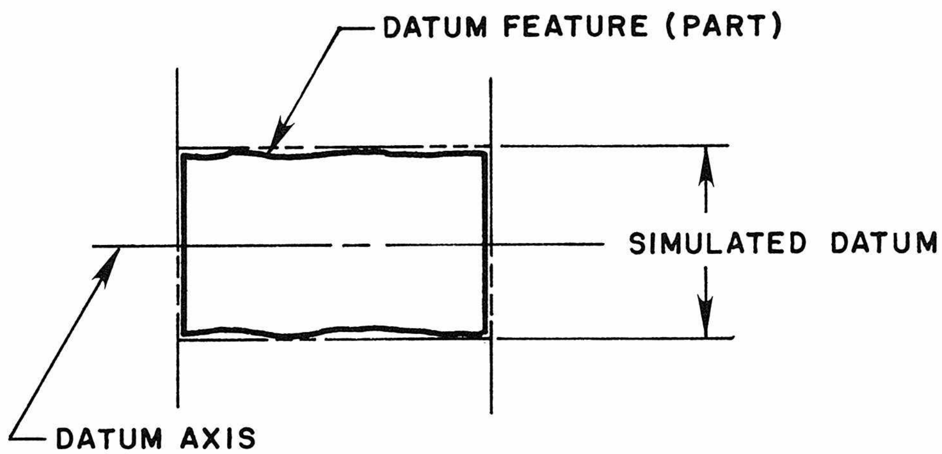
Here datum -A- is the primary datum because the relationship of all features to -A- will extablish perpendicularity. The axis of datum -B- is formed by the intersection of the center planes X and Y.

The X and Y center planes identified above are useful only for describing how the axis is established. The letters X and Y will not actually be placed on your drawing.



Example 3-14

The drawings represented in Examples 3-13 and 3-14 establish datum axis B and the datum feature as displayed in Example 3-15.



Example 3-15

NOTE: For additional practice in handling datum features and to test your skill, see Problem 1, Page 123.

GEOMETRIC DIMENSIONING AND TOLERANCING

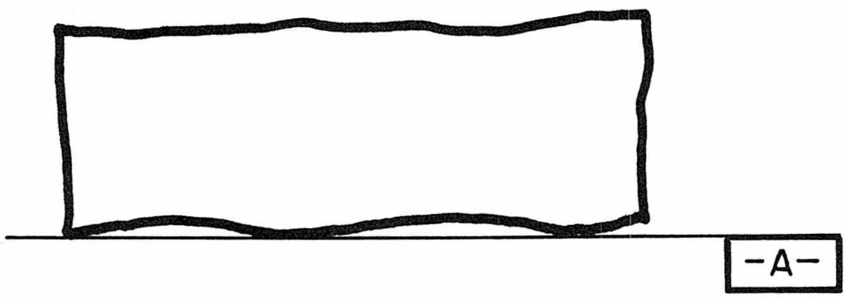
Post Test 3 Name:____________________________

1. Identify the following by placing the related letter with an arrow pointing to the proper location on the illustration below:

   (A) Datum identification symbol.

   (B) Datum feature.

   (C) Datum plane.



2. The primary datum plane requires a minimum of how many points of contact?

   ____________________________________.

3. The secondary datum plane requires a minimum of how many points of contact?

   ____________________________________.

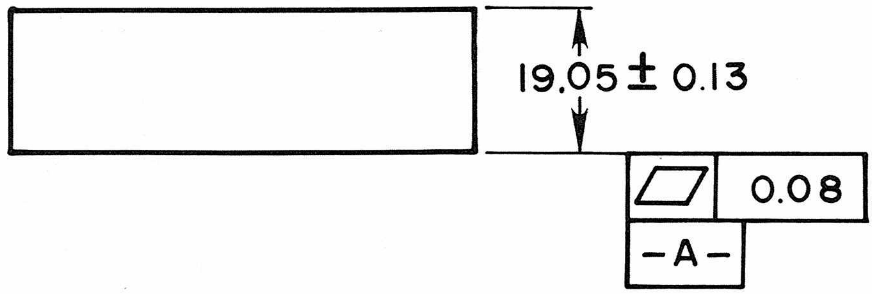
4. The tertiary datum plane requires a minimum of how many points of contact?

   ____________________________________.

5. Define datum, and list four items or types of features that can be used as datums.

Post Test 3

6. In reference to the part example below, give the minimum and maximum sizes and the geometric tolerance: (Show your calculations.)



Minimum size__________________

Maximum size__________________

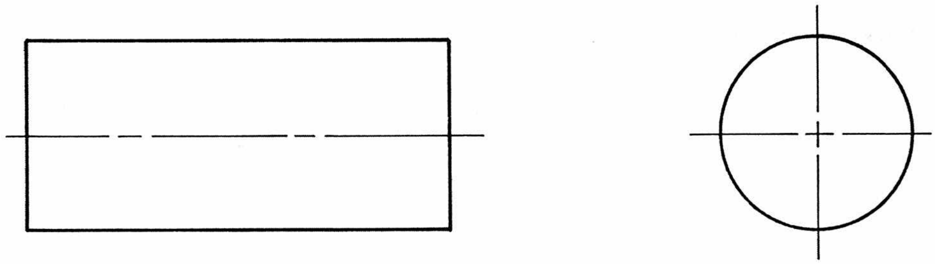
Geometric tolerance__________________

Post Test 3

7. Name the elements of a complete datum frame.

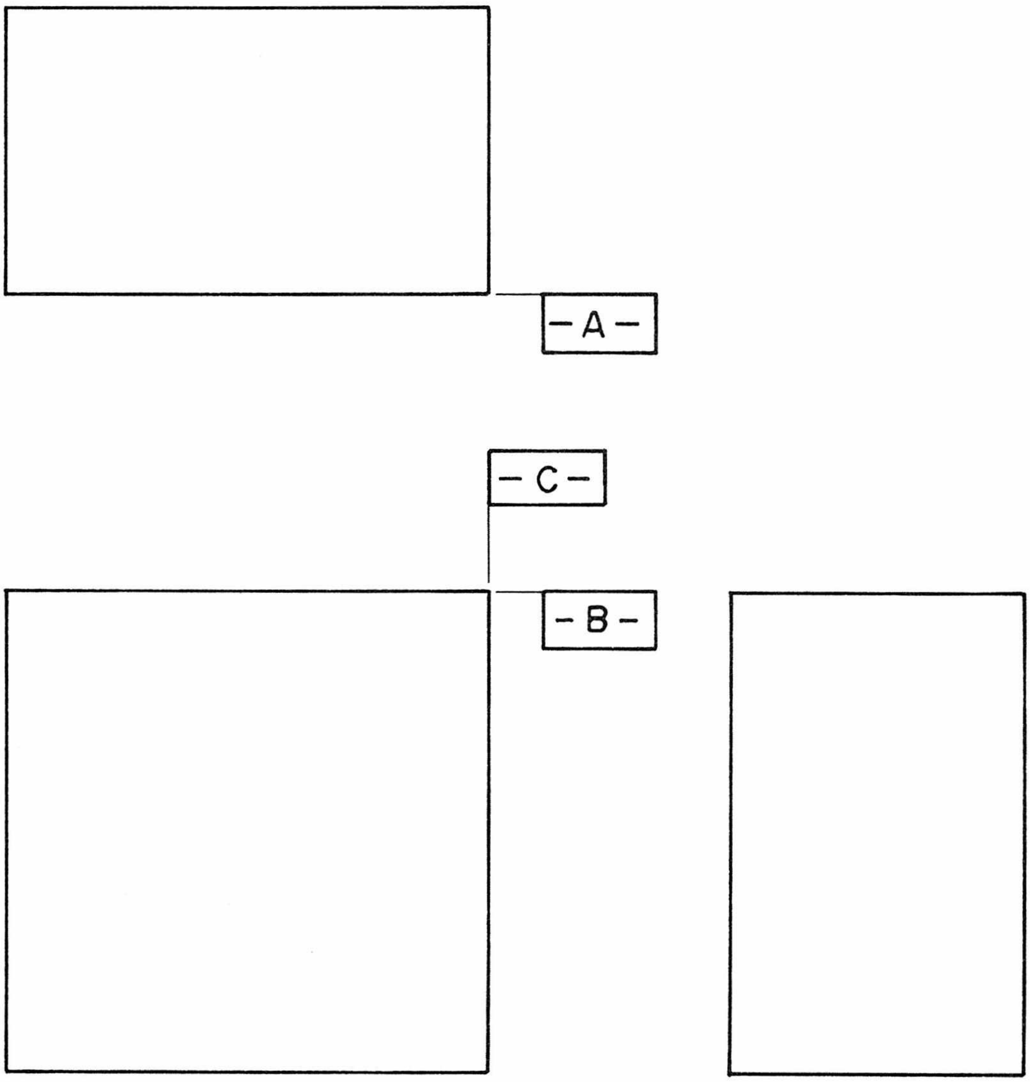
Post Test 3

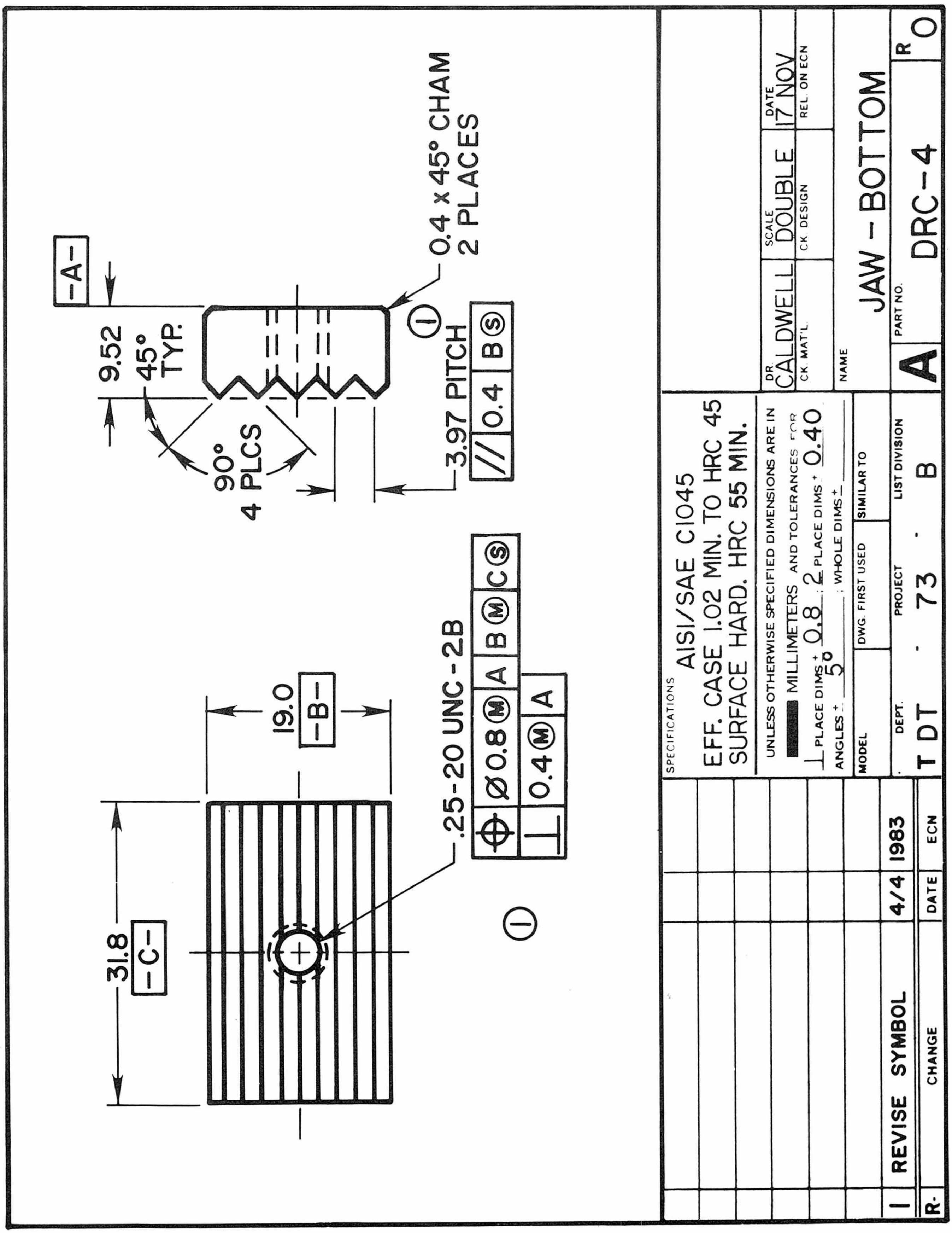
8. Show how the object below would be properly dimensioned so that the center axis is datum -Y-, and one end surface is datum -X-.



Post Test 3

9. Given three views of the object below, locate and label three specific points on datum surface -A-, two on datum surface -B-, and one on datum surface -C-. Your points may be placed anywhere on the respective surfaces. (Draw all symbols properly, do not sketch.)





Typical industry print showing geometric dimensioning and tolerancing.

Chapter 4

MATERIAL CONDITION SYMBOLS

Material Condition Symbols are used in conjunction with feature control symbols to identify how the geometric tolerance zone is effected by the size tolerance. The symbols that are used most commonly are Maximum Material Condition (MMC), or Regardless of Feature Size (RFS).

Least Material Condition (LMC) is used primarily for controlling minimum edge distance.

The discussion that follows will explain how the use of RFS, MMC, or LMC affects the tolerance zone of a feature at any of its possible produced sizes.

MATERIAL CONDITION SYMBOLS

The use of material condition symbols will effect the geometric tolerance in the feature control frame.

These are the material condition symbols Ⓜ Ⓢ Ⓛ

Ⓜ, MMC, MAXIMUM MATERIAL CONDITION

Maximum Material Condition indicates the most material possible in an object.

Ⓜ used in a feature control symbol means that the specified geometric tolerance will only be held when the part is at MMC.

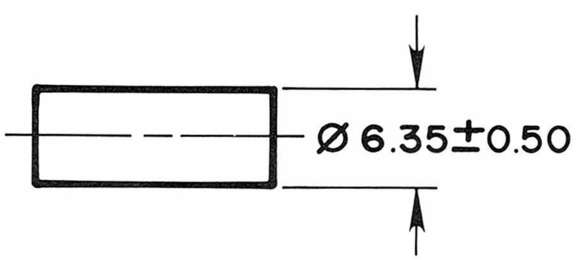
⌖ Position is the only geometric characteristic that must have a material condition symbol with the given geometric tolerance, either Ⓜ, Ⓢ or Ⓛ

Ⓢ, RFS, REGARDLESS OF FEATURE SIZE

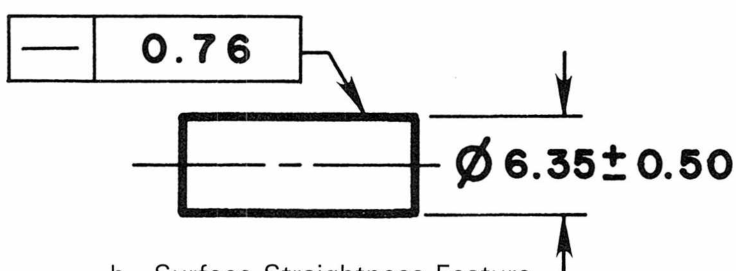
Regardless of feature size, RFS is implied when any of the geometric characteristics shown in Example 4-1 are given in the feature control frame.

Example 4-1

Example 4-2a shows an object with a conventional dimension and Example 4-2b shows a feature control frame added.



a. Conventional Tolerance



b. Surface Straightness Feature Control Frame. RFS Implied

Example 4-2

Using conventional tolerancing, the possible range of produced sizes is any size between 6.85 MMC to 5.85 LMC, as you can see in Example 4-2a. When the feature control frame is added to the drawing as in Example 4-2b, control is now applied to the geometric shape. The feature control frame connected to the surface of the object controls surface straightness of the object and RFS is assumed.

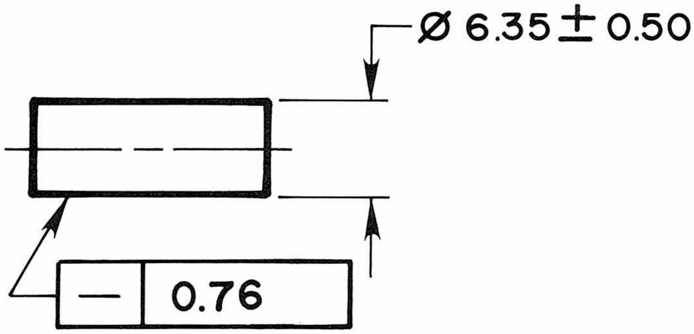
Perfect form or zero geometric tolerance is required at MMC for surface straightness, as shown in Example 4-4. Now, as the produced sizes depart from MMC, the geometric tolerance increases until the amount of departure from MMC equals the geometric tolerance given in the feature control frame. A list of produced sizes and related geometric tolerances for the object in Example 4-2b is as follows:

	MMC				LMC
Produced Sizes	6.85	6.60	6.35	6.10	5.85
Geometric Tolerance	0.00	0.25	0.50	0.76	0.76

SURFACE CONTROL, REGARDLESS OF FEATURE SIZE

Ⓢ assumed, material condition symbol not required for geometric characteristics shown in Example 4-1, RFS is assumed. Some companies, for clarity, may prefer to place the material condition symbol in the feature control frame.

Surface straightness should not use an MMC material condition symbol and a diameter symbol should not be specified as shown in Example 4-3.



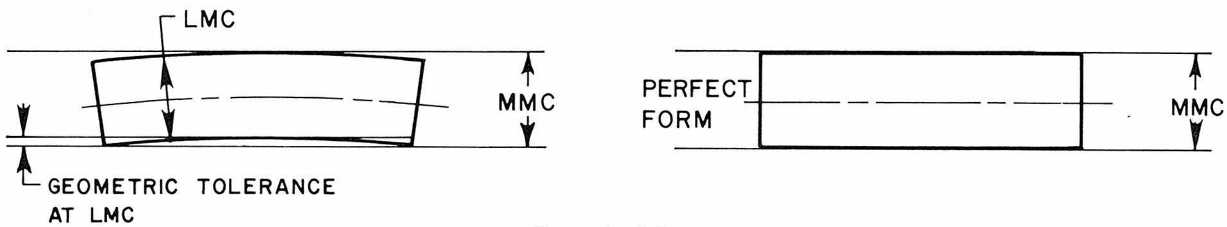
Example 4-3

With Ⓢ being used, the geometric tolerance effects the tolerance zone as shown to the right. RFS is implied for tolerance of form, and the symbol is usually not used. The inclusion of the Ⓢ here may be a company preference but not required by the ANSI standard.

Now, look at what happens to the maximum out-of-straightness.

Produced Sizes	Maximum Out-of-Straightness
MMC 6.85	0.00
6.60	0.25
6.35	0.50
6.10	0.76
LMC 5.85	0.76

Example 4-4 shows how the use of a material condition symbol, regardless of feature size, affects the geometric tolerance within the maximum material condition envelope.

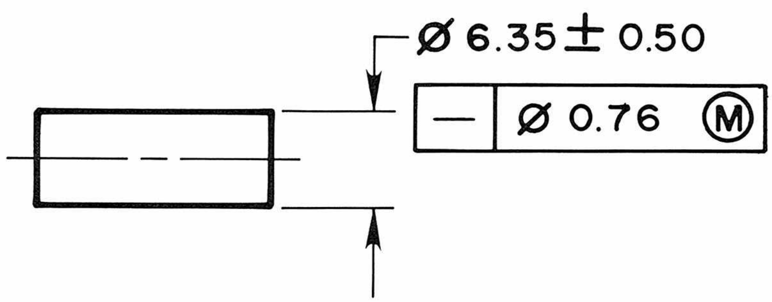


Example 4-4

AXIS CONTROL AT MAXIMUM MATERIAL CONDITION AND REGARDLESS OF FEATURE SIZE

Ⓜ **Used** Axis control

For axis straightness control (and use of MMC or RFS with straightness) the feature control frame must be associated with the dimension of the feature and include a diameter symbol for cylindrical features as shown in Example 4-5.



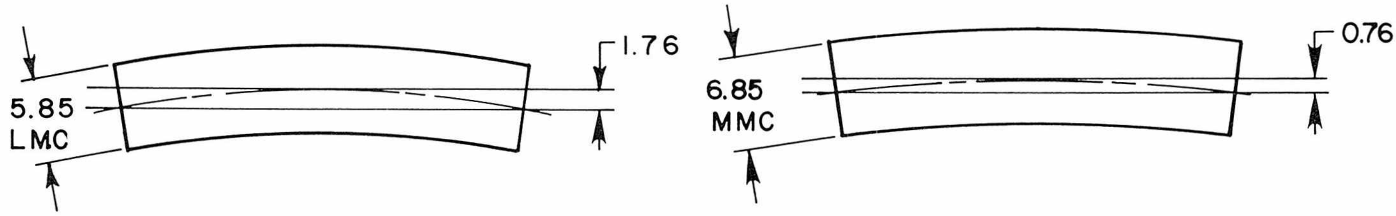
Example 4-5

Axis control, MMC, Material Condition Symbol

The chart below shows how the geometric tolerance, shown in Example 4-5, changes at several produced sizes with MMC used as a material condition symbol. With Ⓜ being used, the geometric tolerance is the same as specified in the feature control frame at the MMC produced size. The maximum geometric tolerance gets larger as the produced size departs from MMC. This difference in geometric tolerance is equal to the amount of departure from the MMC size.

Produced Sizes	Ⓜ used Maximum Out-of-Straightness of axis.
MMC 6.85	0.76
6.60	1.01
6.35	1.26
6.10	1.51
LMC 5.85	1.76

Example 4-6 shows how the use of a maximum material condition symbol effects the geometric tolerance of an object's axis.



Example 4-6

When MMC is not specified then RFS is implied. When this situation occurs, the geometric tolerance at various produced sizes remains the same as given in the feature control frame. Perfect form at MMC is not required for axis straightness as it was for surface straightness.

Ⓢ assumed if MMC not used.

Produced Sizes	Maximum Out-of-Straightness
MMC 6.85	0.76
6.60	0.76
6.35	0.76
6.10	0.76
LMC 5.85	0.76

With Ⓢ being used, the geometric tolerance is the same at any produced size.

LEAST MATERIAL CONDITION

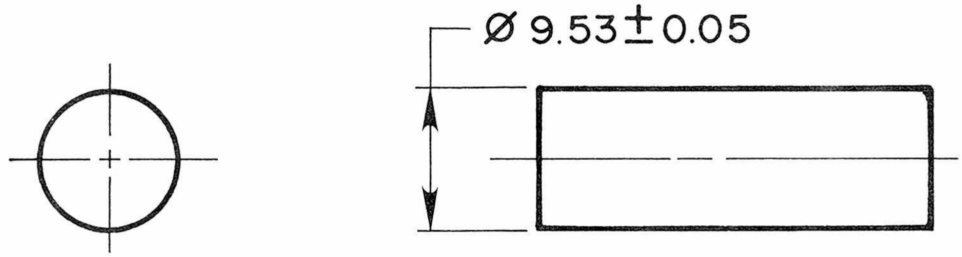
Ⓛ LMC is the condition where the feature of size contains the least material. For example, maximum hole diameter and minimum shaft diameter—the opposite of MMC. When LMC is used in the feature control frame, the given geometric tolerance is held at the LMC produced size. When the produced size departs from LMC toward MMC, the geometric tolerance changes equal to the difference from LMC. The maximum amount of departure is at the MMC produced size.

LEAST MATERIAL CONDITION IS GENERALLY USED FOR CONTROLLING MINIMUM EDGE DISTANCE.

GEOMETRIC DIMENSIONING AND TOLERANCING

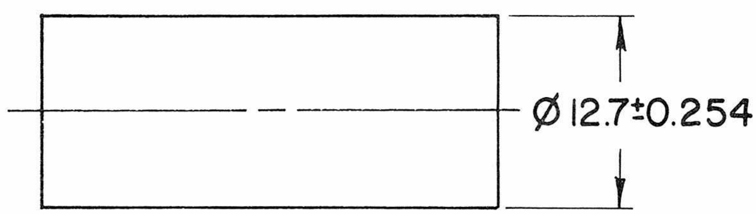
Post Test 4 Name: ____________________

1. Complete the following table for the callouts indicated at the actual feature sizes given. Assume axis control.



Actual Feature Size	— Ø 0.025	— Ø 0.025 M
9.58		
9.55		
9.525		
9.50		
9.49		
9.48		

2. Complete the following table for the callouts indicated at the actual feature sizes given. Assume surface control.



Actual Feature Size	— 0.127
12.954	
12.903	
12.852	
12.827	
12.800	
12.750	
12.700	
12.650	
12.598	
12.548	
12.498	
12.446	

Post Test 4

3. What is the symbol for RFS?

RFS is the abbreviation for what?

What is the definition of RFS?

4. What is the symbol for MMC?

MMC is the abbreviation for what?

What is the definition of MMC?

5. How does the use of RFS affect a geometric tolerance?

6. How does the use of MMC affect a geometric tolerance?

7. What is the symbol for LMC?

LMC is the abbreviation for what?

What is the definition of LMC?

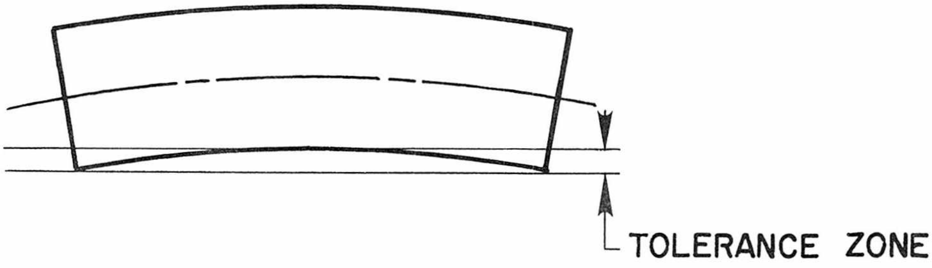
Chapter 5

GEOMETRIC CHARACTERISTICS

This chapter will show you the relationship that must exist between size limits and geometric form. Normal size dimensioning and surface texture identification provides a certain amount of control upon the shape of an object. The application of geometric tolerance to these considerations is an aid to design when more accurate designations of form are needed for functional interchangeability of parts. Geometric tolerances specify how much actual feature variation is permitted. This specification must be applied in relationship to the size.

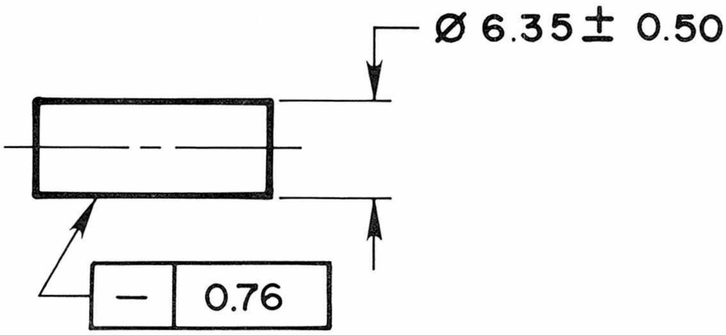
After careful study, you will see, that certain geometric tolerances, due to their characteristics, include elements of other geometric tolerances. For example, cylindricity controls circularity, but circularity does not control cylindricity. Also parallelism could control flatness and straightness. Now you will be able to apply the symbolism studied in Chapter 2 to actual situations in this chapter. Remember to draw all symbols and notes using full scale realistic drafting techniques, unless your instructor specifies differently.

STRAIGHTNESS



Example 5-1

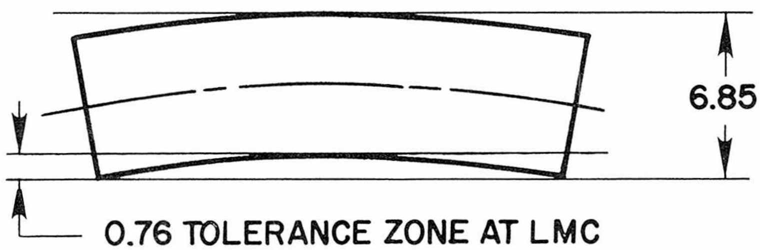
Example 5-1 is an exaggerated representation of what really happens when a surface straightness tolerance is applied to an object. The object must be straight within the tolerance zone specified. Note Example 5-2 how a straightness tolerance is applied on a drawing to control surface straightness.



Example 5-2

SURFACE STRAIGHTNESS

Example 5-2 really means that the object must be produced as shown in Example 5-3. The feature may not exceed the MMC envelope.



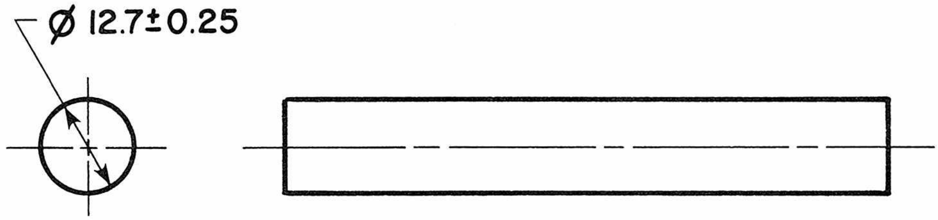
Example 5-3

Remember, straightness implies RFS as does the other form characteristics. This is what happens at several produced sizes.

Possible Produced Sizes	Maximum Out-of-Straightness
6.85	0.00
6.45	0.40
6.30	0.55
6.09	0.76
6.05	0.76
6.00	0.76
5.90	0.76
5.85	0.76

Try this:

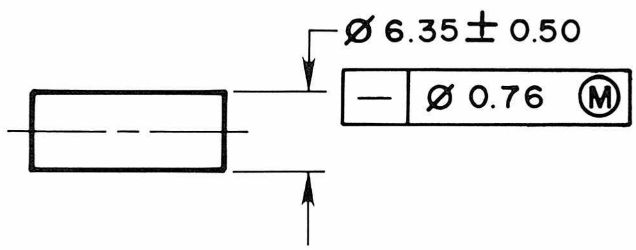
Provide a callout so that the following object will be straight (surface) within 0.127. Use proper size symbolism. Do not sketch.



What is the maximum out-of-(surface) straightness acceptable at the following produced sizes?

Possible Produced Sizes	Maximum Out-of-Straightness
12.95	
12.90	
12.85	
12.80	
12.75	
12.70	
12.65	
12.60	
12.55	
12.50	
12.45	

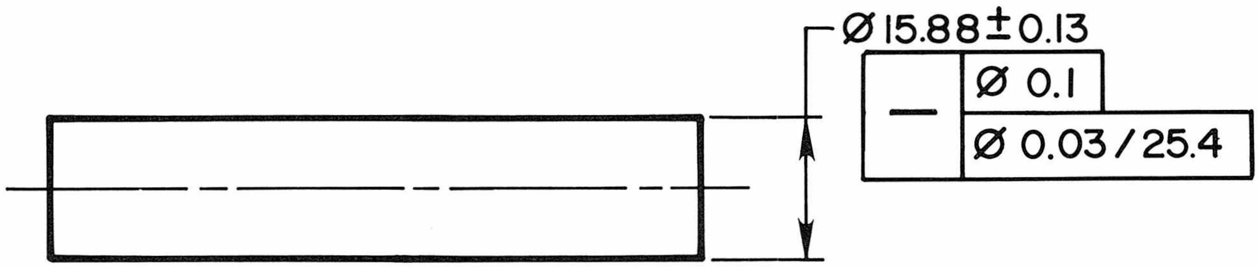
Axis straightness is represented on a drawing as shown in Example 5-4. Axis straightness may violate perfect form at MMC. See Example 4-5, pages 37 and 38, for review.



Example 5-4

AXIS STRAIGHTNESS

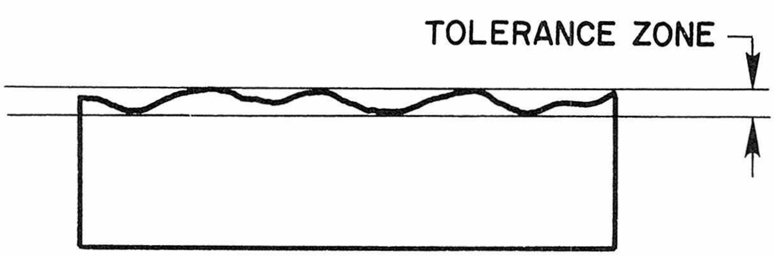
Straightness per unit of measure may often be applied to an object. When this is done a total straightness may also be included. For example, we can apply a straightness per unit tolerance, but we may also provide a total tolerance over the length of the part. Look at Example 5-4 for a sample drawing with unit straightness applied.



Example 5-4

Be careful in using unit straightness as it could cause waviness in a part.

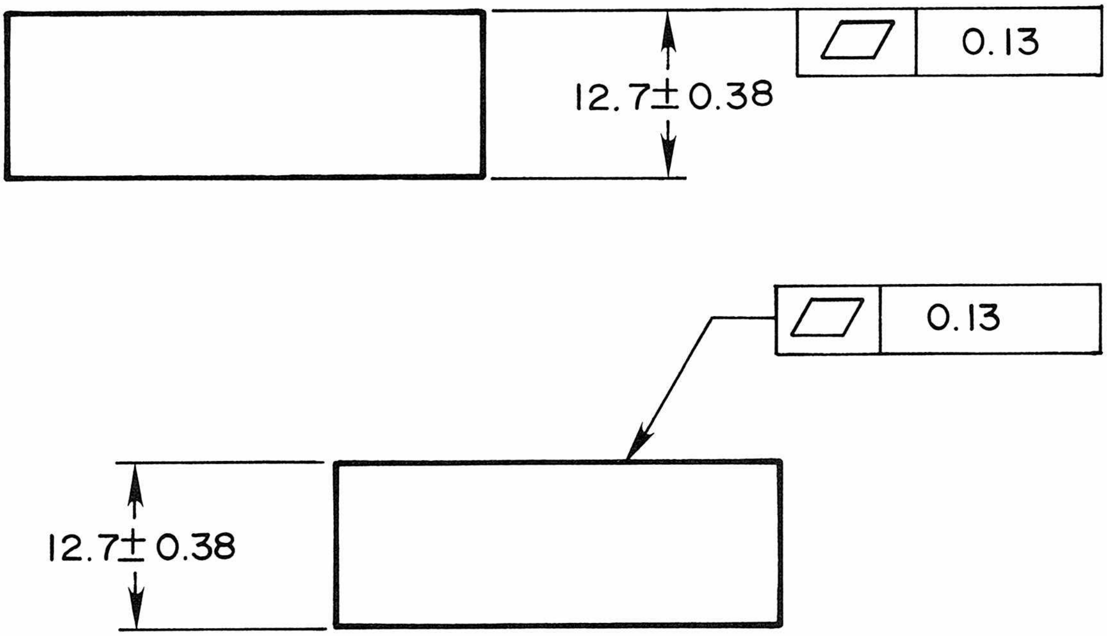
FLATNESS



Example 5-5

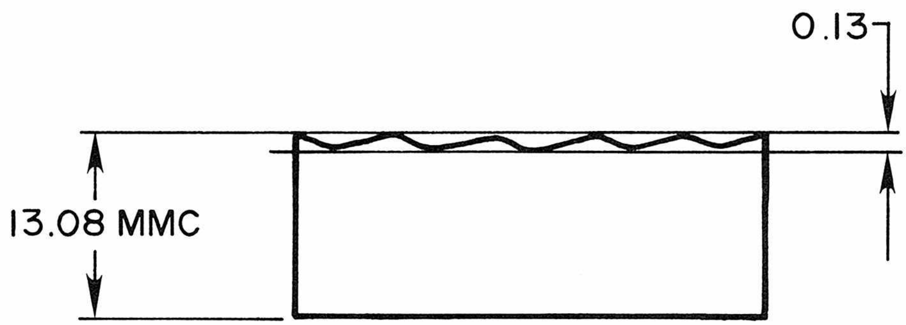
Example 5-5 is an exaggerated demonstration of what a flatness tolerance zone means. All of the points of the surface must be "within" the limits of this zone.

The smaller the zone the flatter the surface will be. Example 5-6 shows how an object could be dimensioned to provide for a flatness callout.



Example 5-6

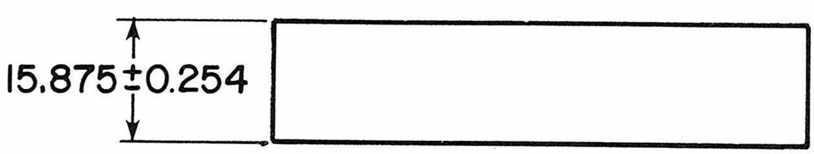
The meaning of the flatness tolerance applied to Example 5-6 is demonstrated in Example 5-7.



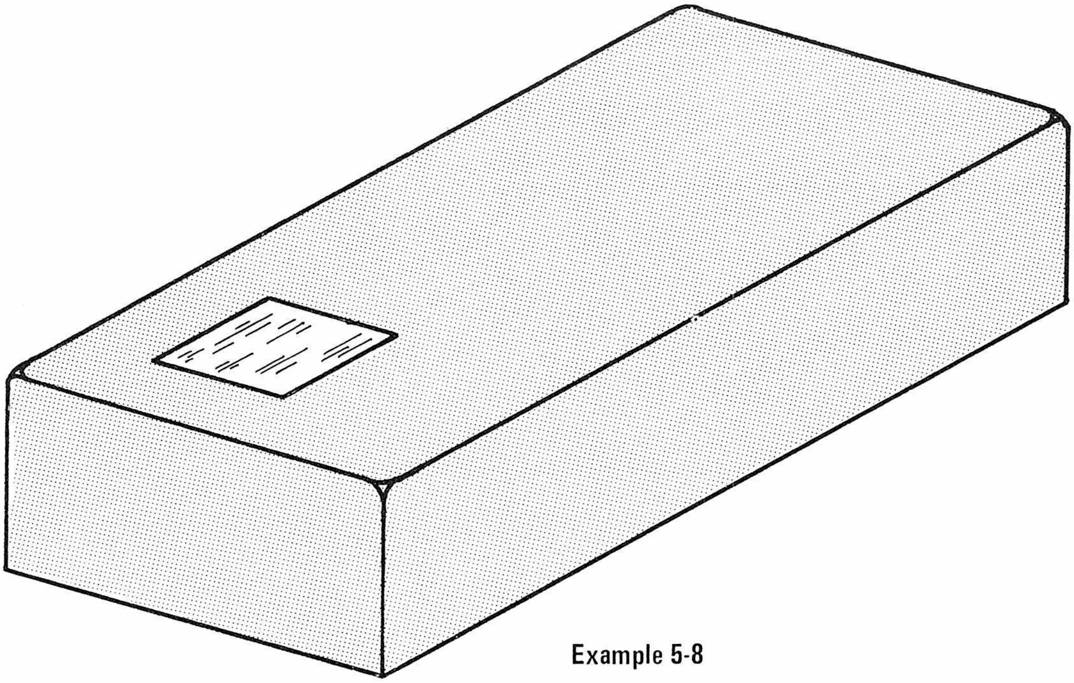
Example 5-7

You try this:

Provide a 0.025 flatness tolerance zone to the top surface of the object below.

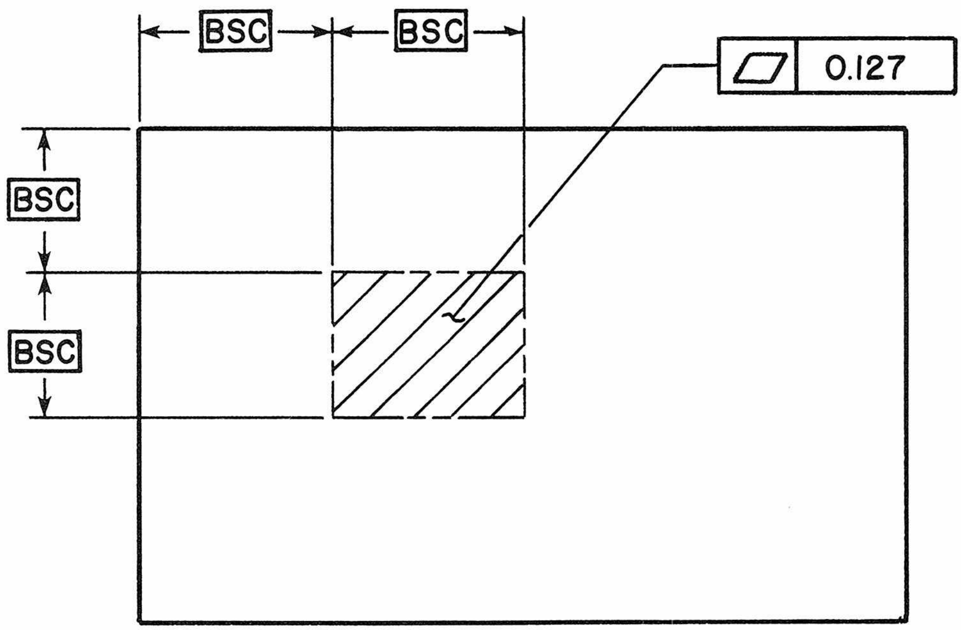


At times it may be necessary to provide a flatness callout for only a specific area of a large surface as indicated in Example 5-8.



Example 5-8

This procedure may be used when a large cast surface must be flat, but it may be possible to finish a small area rather than an expensive operation of finishing the entire surface. Example 5-9 shows how this can be accomplished on a drawing.

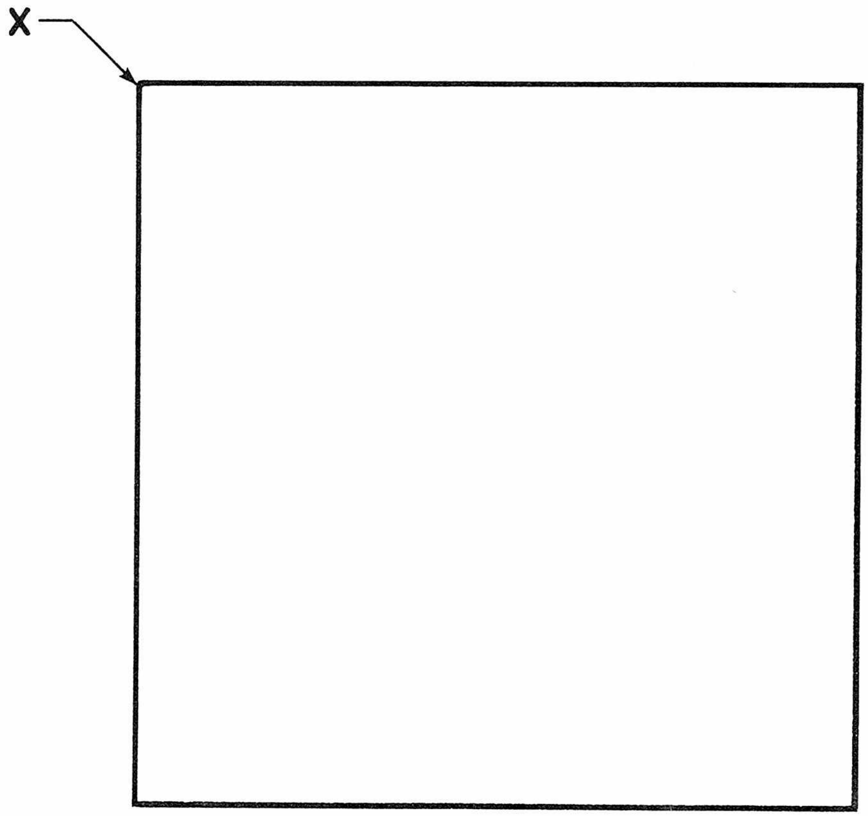


Example 5-9

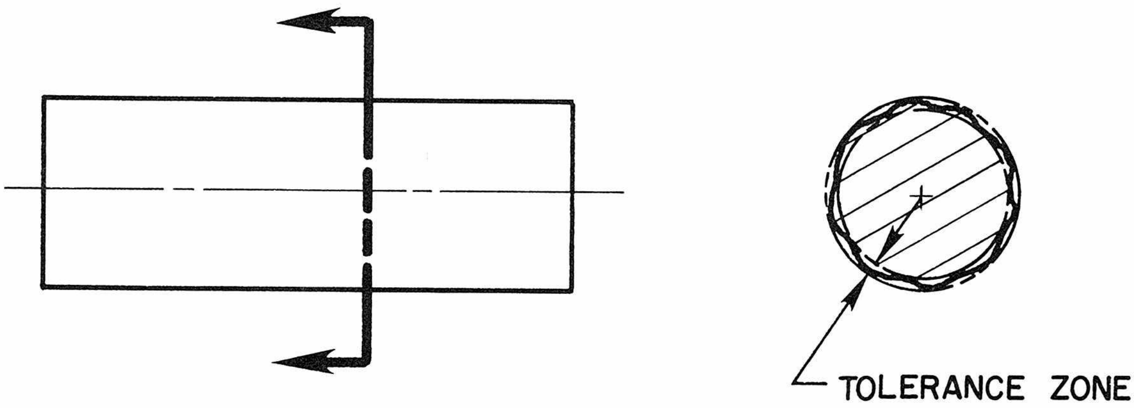
The specific area is outlined with phantom lines. Then it is section lined. Locate the specific area with basic or ± dimensions. Finally, identify the specific area with a feature control frame similar to the method used in Example 5-9.

Try this:

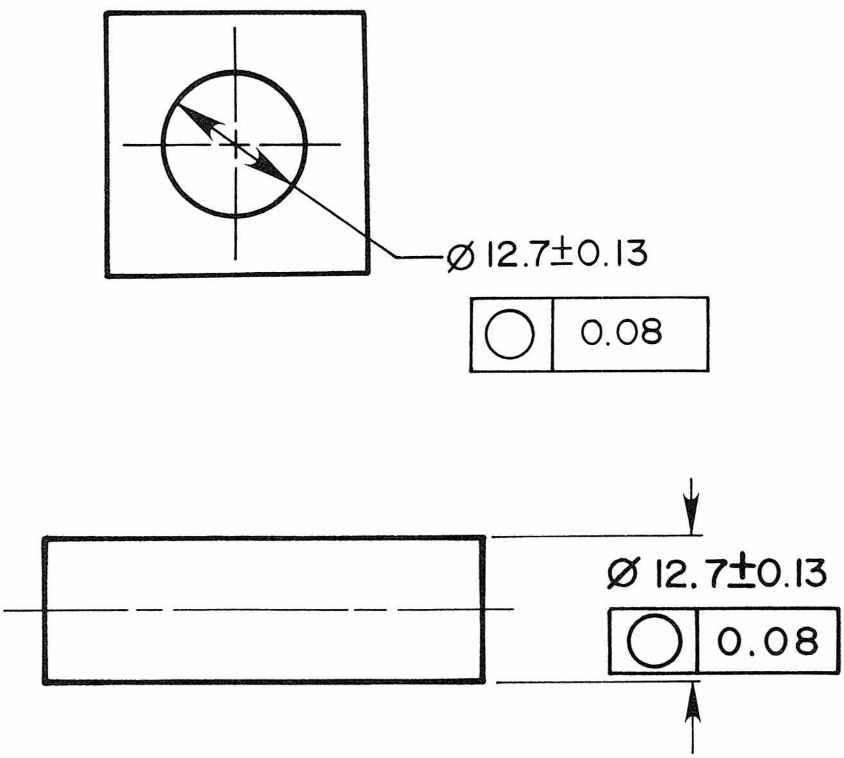
Given the object below, provide a flatness callout with a tolerance zone of 0.076 to a specific area that is 25.4 mm square and 31.75 mm both ways from the corner marked "X." (Use proper techniques.)



○ CIRCULARITY

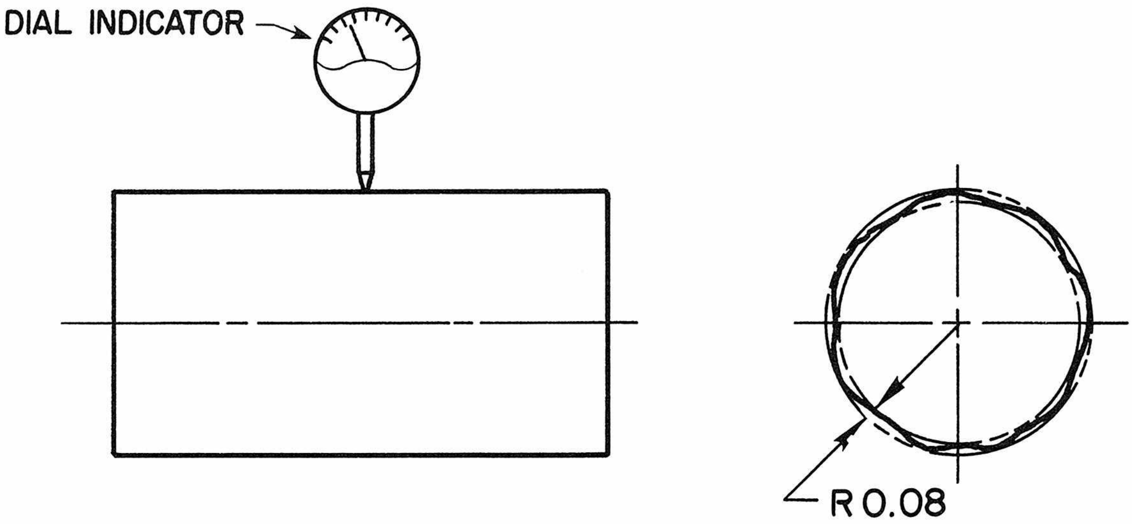


The exaggerated example above shows that circularity is actually a geometric characteristic that is a tolerance zone at any given cross-sectional location. The tolerance only applies to one sectional element at one time. A circularity tolerance can be shown on an object as in Example 5-10. Circularity should be calculated with respect to a radius tolerance zone.



Example 5-10

Example 5-11 is a display of what the circularity tolerance shown in Example 5-10 means.

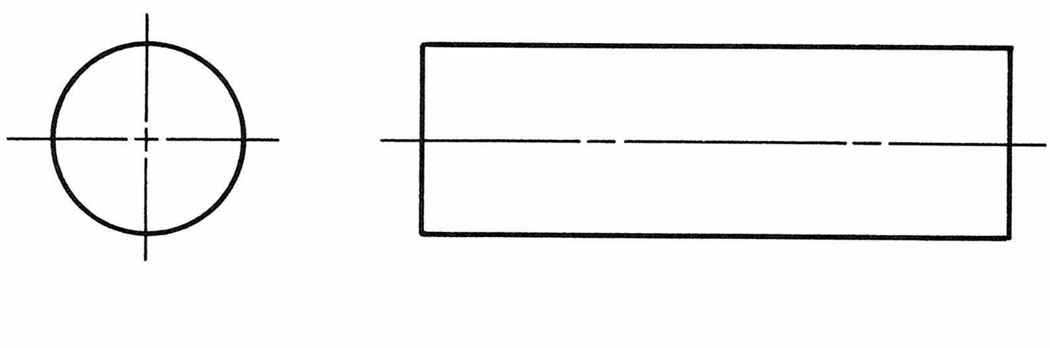


Example 5-11

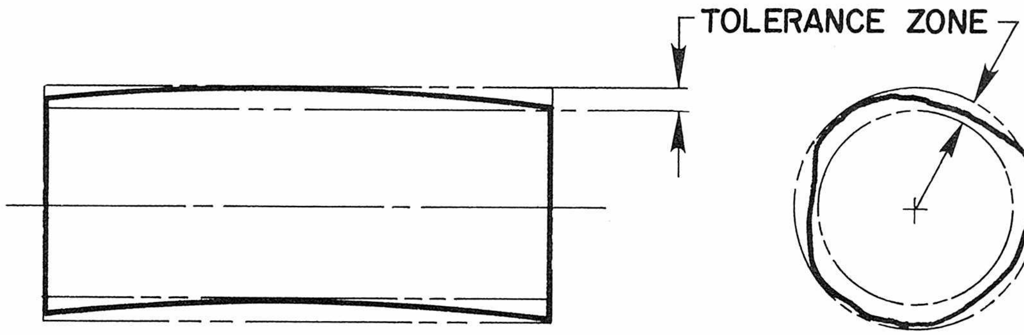
Remember the zone descriptor is on radius or equal to the total movement of the dial indicator.

Do this:

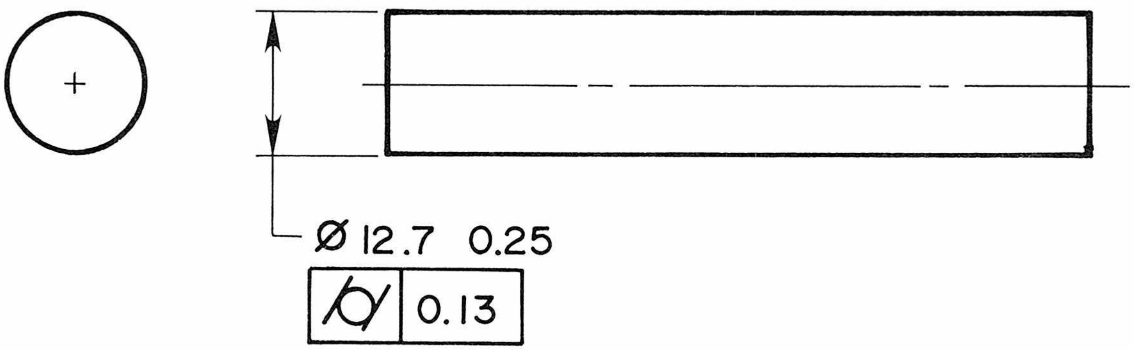
Provide a circularity callout with a 0.10 tolerance zone to the object below.

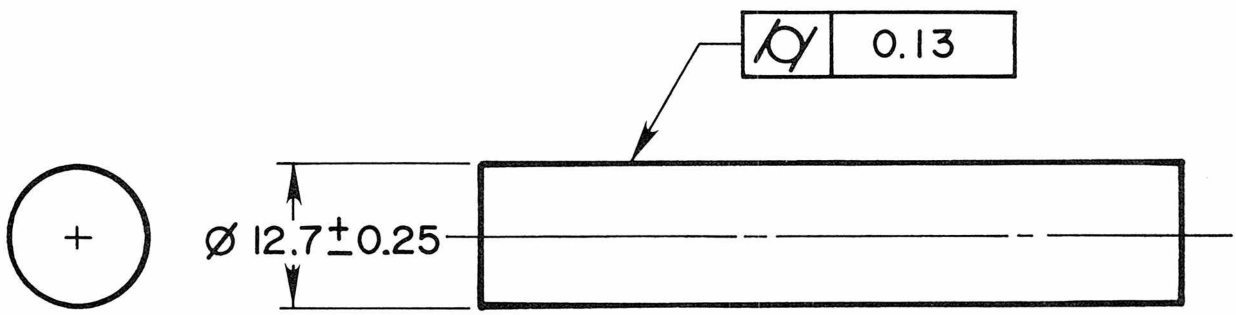


CYLINDRICITY



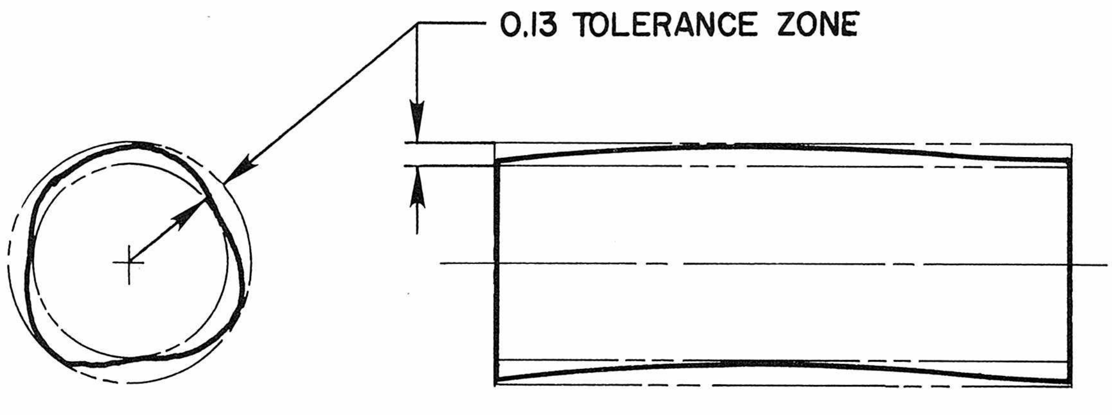
Cylindricity is identified by a radius tolerance zone that establishes two perfectly concentric cylinders. The object can be produced anywhere within this zone as shown in the example above. Now look at Example 5-12 below and see how cylindricity is dimensioned on an object.





Example 5-12

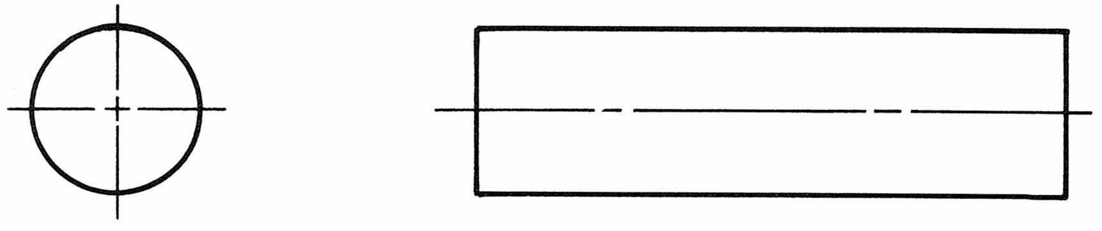
In Example 5-13 you will see what is meant by the objects dimensioned in Example 5-12.



Example 5-13

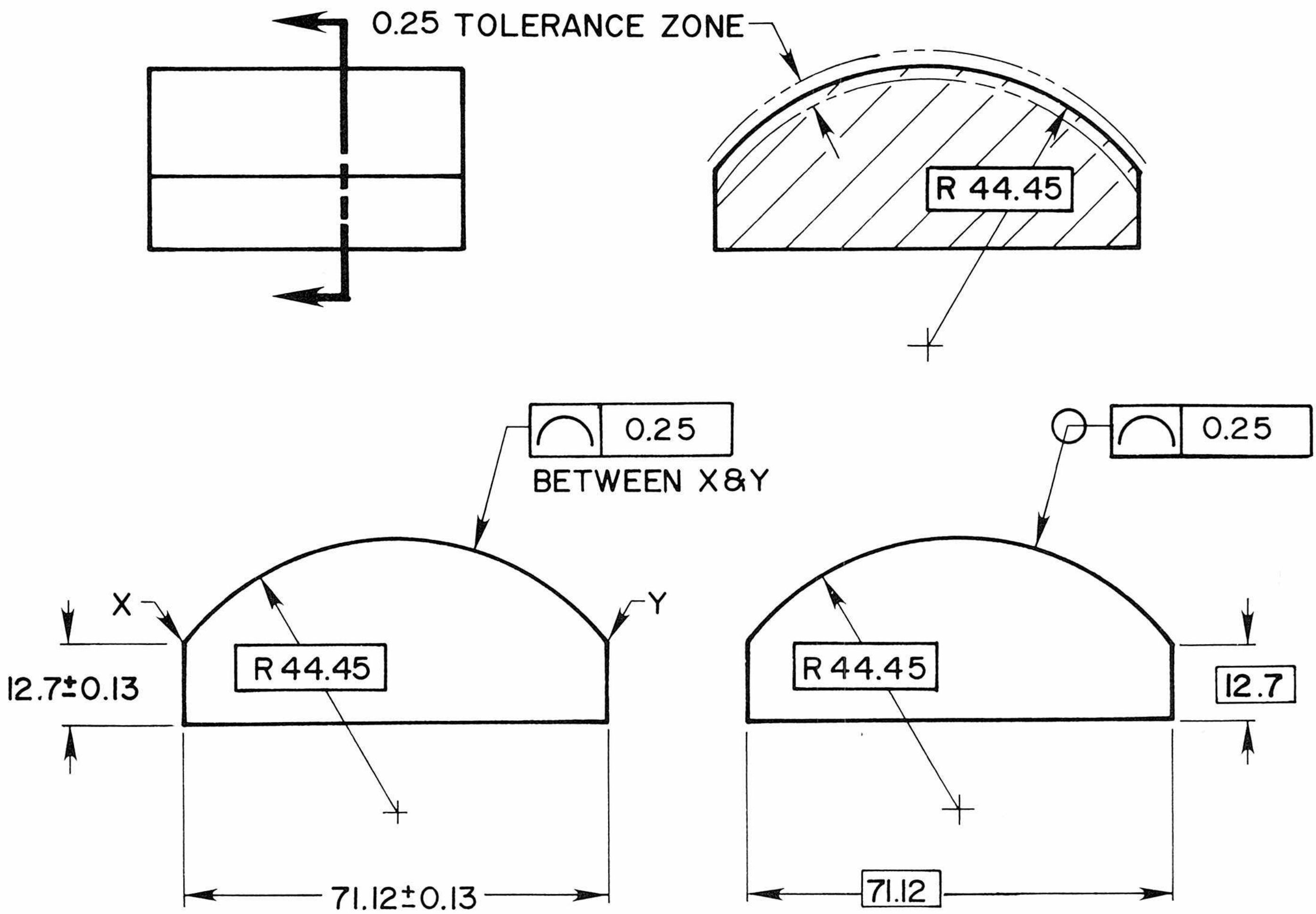
Try this:

Provide a 0.254 cylindricity tolerance to the 20.955 diameter object below.



PROFILE

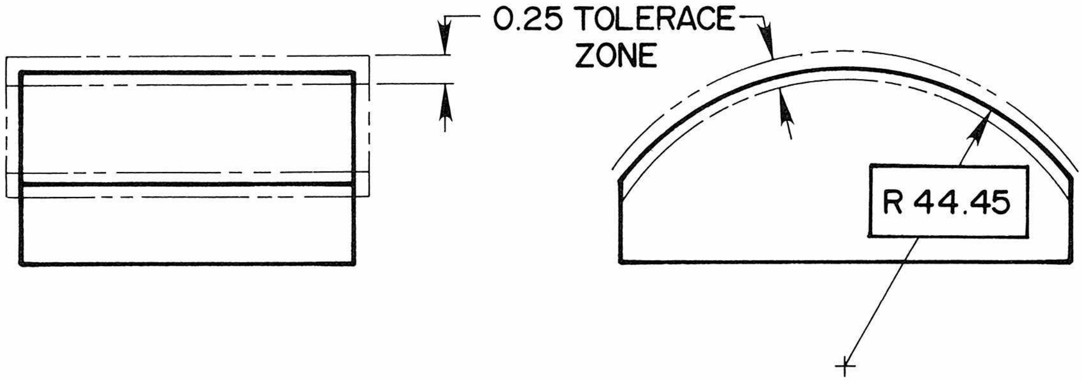
Profile of a line is used for any line element of a surface that does not require control of the entire length of the surface, or to objects that have different cross-sectional dimensions throughout their length. Example 5-14 shows that the profile of a line is any line of a surface (represented by the cutting plane line). The extreme ends of this line should be identified with letters such as "between X and Y." If the tolerance zone goes all the way around the object, then "all around" symbol, ( ) should be specified. Example 5-14 also shows how this is accomplished on a drawing. The geometric profile tolerance must fall within the limits of the specified tolerance zone for size.

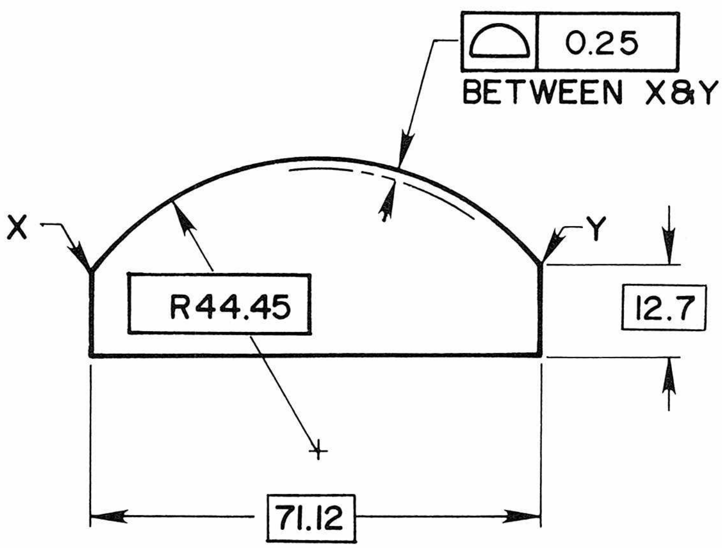


Example 5-14

Examples using bilateral tolerance.

⌓ Profile of a surface is different in that the entire length of the surface of the object will be controlled by the geometric tolerance zone. Again, you should specify between two given points or all around the object. Example 5-15 shows what profile of a surface really means.





Example 5-15

Example using unilateral tolerance.

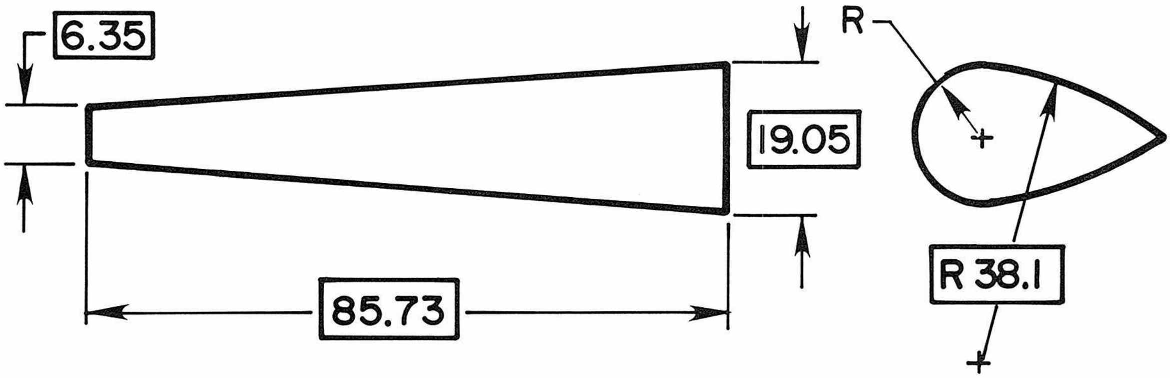
Profile may also have a unilateral tolerance as shown here with the tolerance to the inside. An outside tolerance may also be specified by placing the phantom line outside of the object.

Usually the profile of a surface would have a datum reference to provide orientation of the tolerance zone; however, profile of a line may relate to a datum, but not when only cross-sectional control is required.

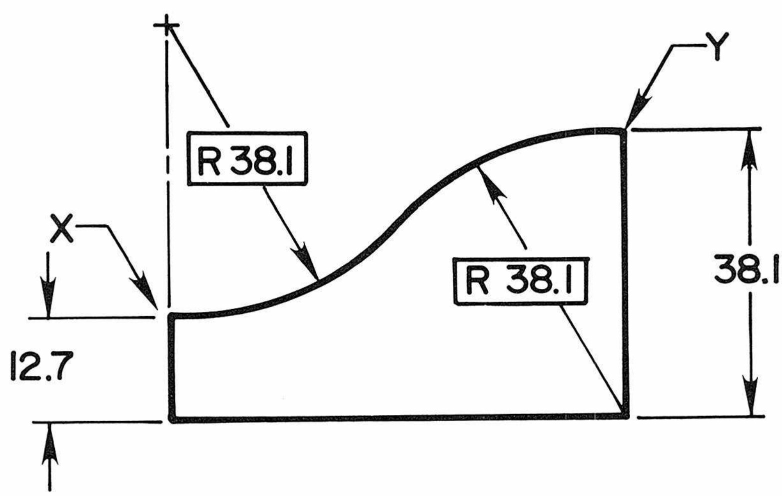
Coplaner surfaces are two or more surfaces on a part that are on the same plane. To control the profile of these surfaces as a single surface place a phantom line between the surfaces in the view where the required surfaces appear as lines. Connect a leader to the phantom line with a profile feature control frame. Add the words 2 SURFACES below the feature control frame.

Try these problems:

Provide a line profile with a .005 tolerance all around the object below.

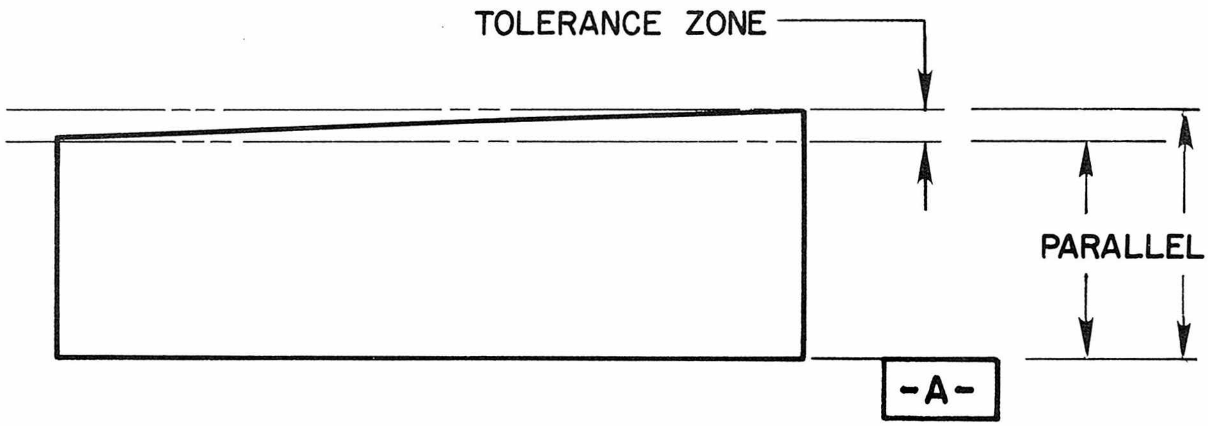


Provide a callout profile to the surface between points X and Y below. Allow a tolerance of 0.20.



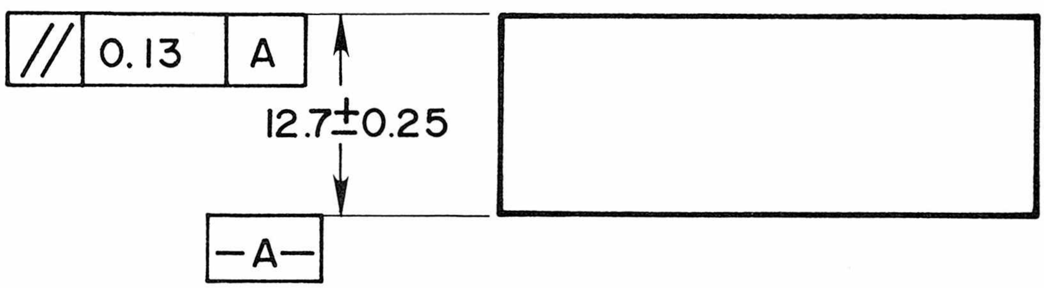
// PARALLELISM

Any feature of an object can be specified as being parallel to any given parallel datum. The demonstration in Example 5-16 shows that a parallelism callout indicates that a surface must be within the bounds of the zone that is parallel to the datum.



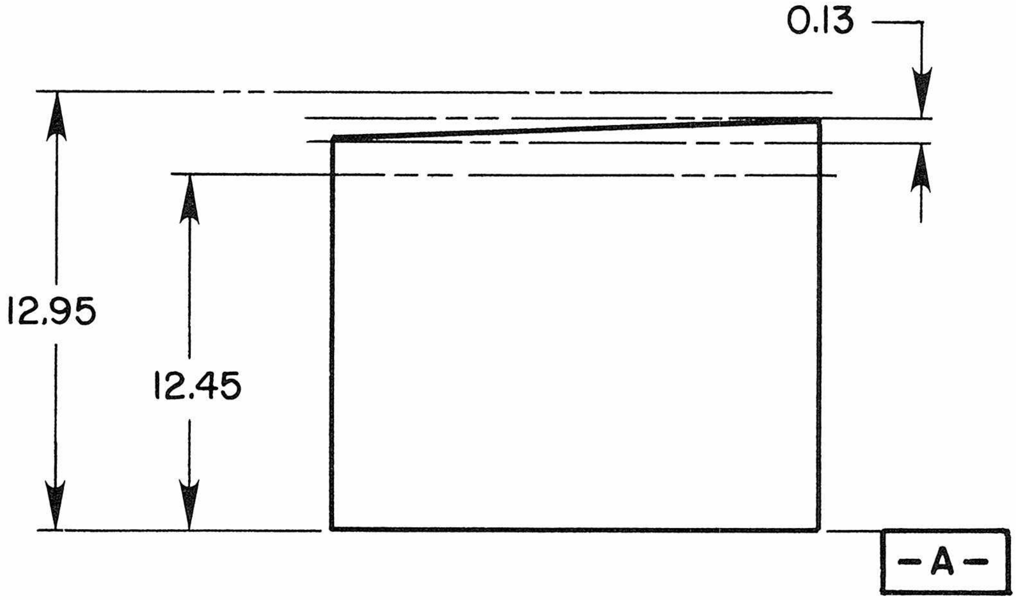
Example 5-16

Example 5-17 shows how parallelism is dimensioned on a drawing.



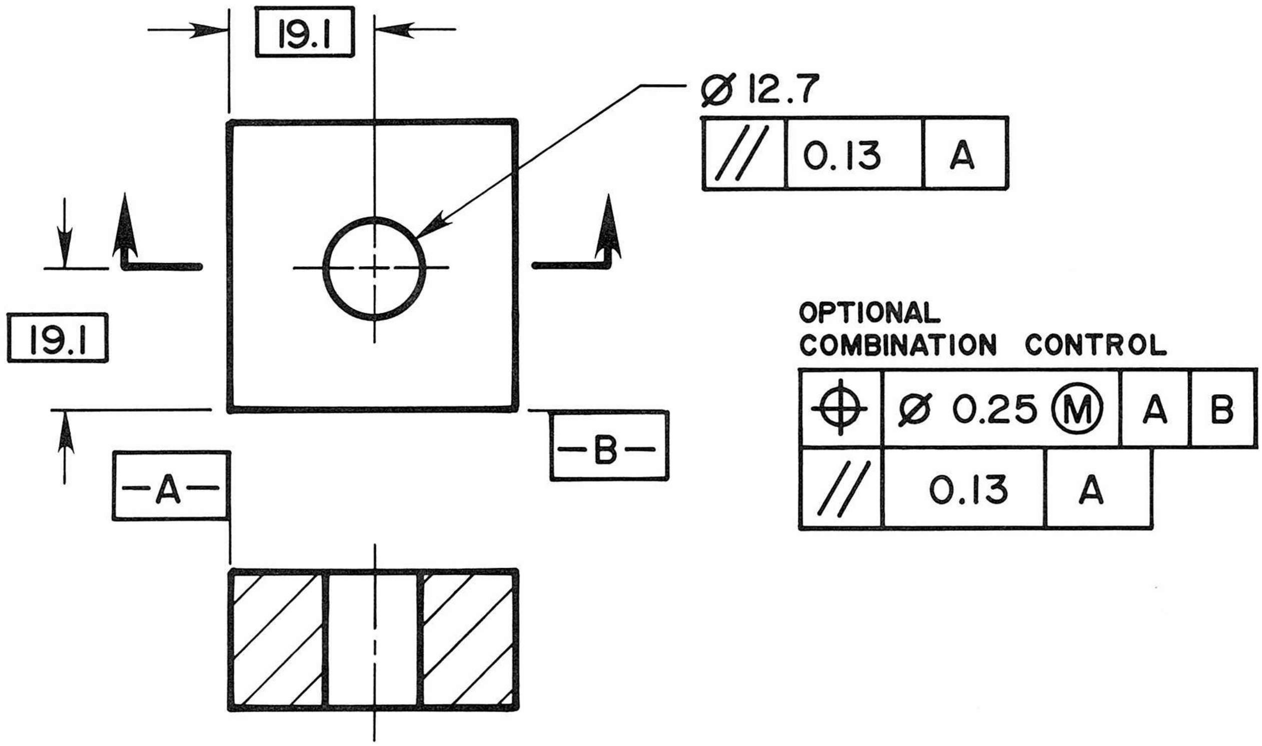
Example 5-17

Example 5-17 means that the parallelism tolerance must be held within the limits of the size dimensional tolerance. Example 5-18 shows an example of a surface which is within a tolerance zone.

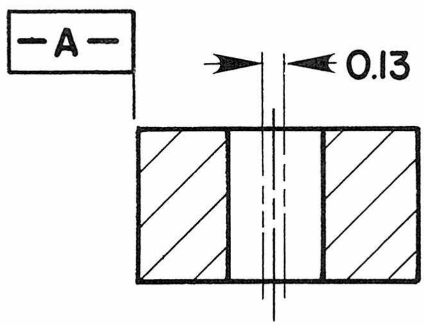


Example 5-18

Example 5-19 shows another sample of parallelism.



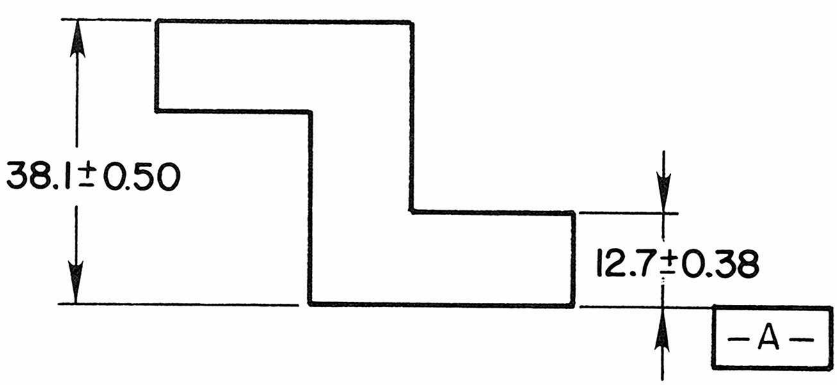
This Means:



Example 5-19

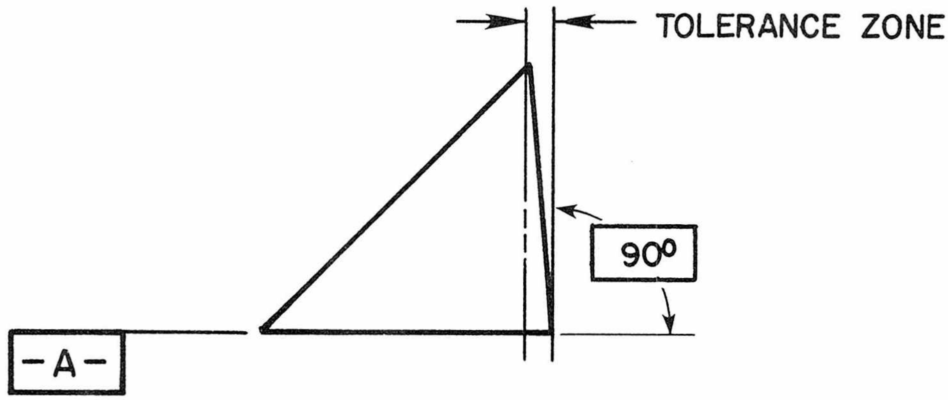
Try this:

Dimension the object below so that the top surface of the object will be parallel to datum -A- by a tolerance of 0.2.



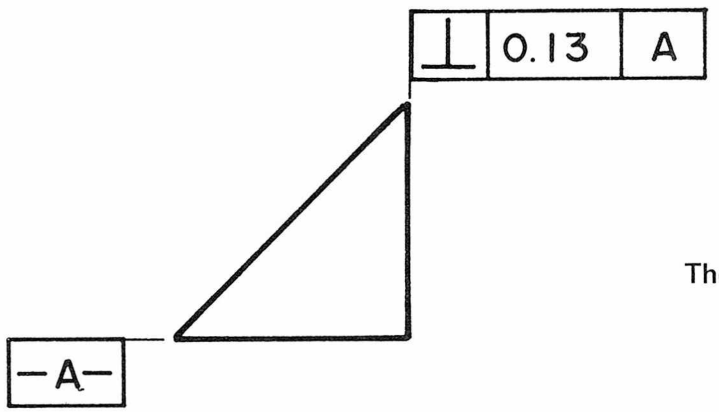
⊥

PERPENDICULARITY

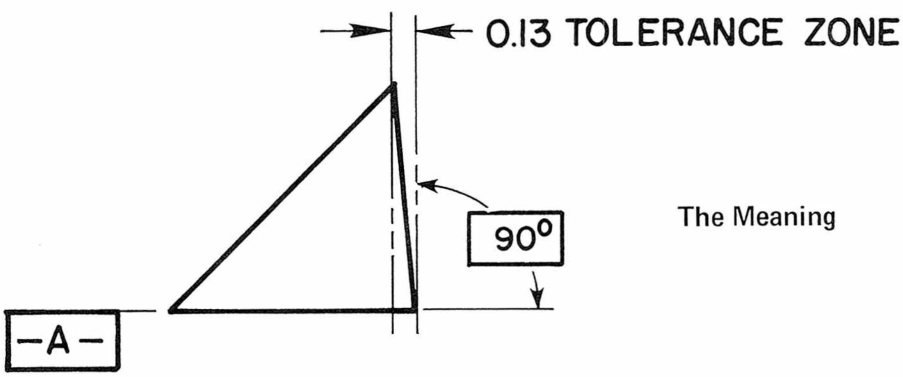


Perpendicularity is established by a given tolerance zone that is 90° to a given datum.

Example 5-20 shows how perpendicularity can be shown on a drawing, and also what the drawing means.



The Drawing

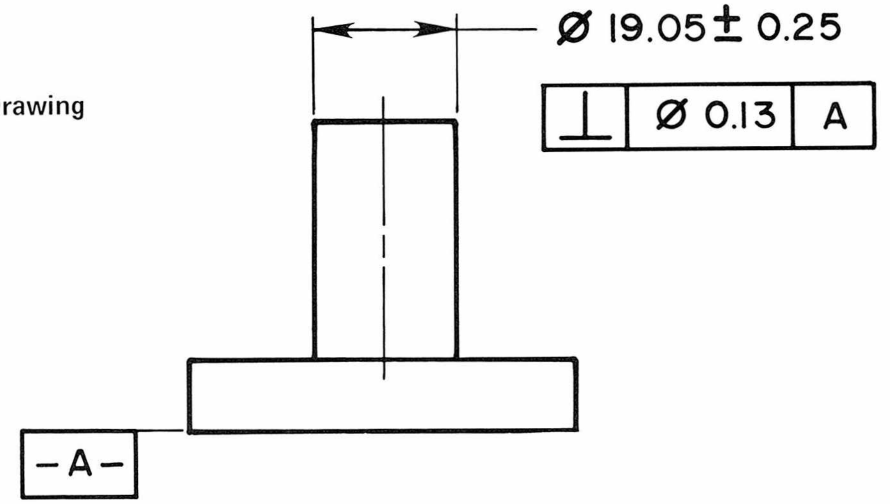


The Meaning

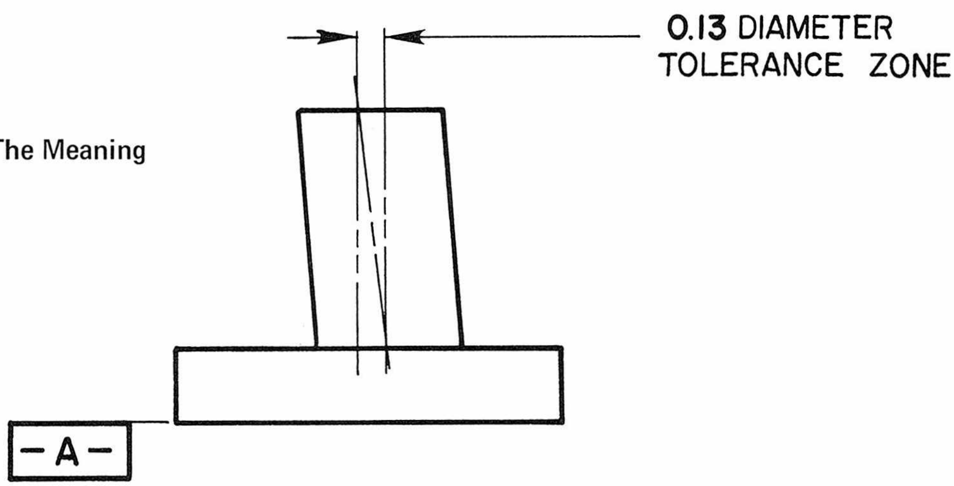
Example 5-20

Another situation that requires perpendicularity could be a symmetrical object such as Example 5-21.

The Drawing



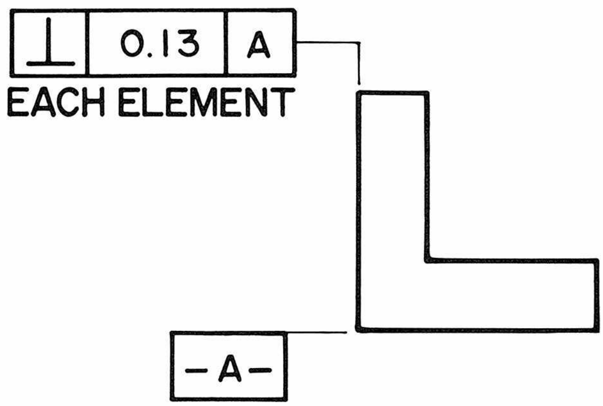
The Meaning



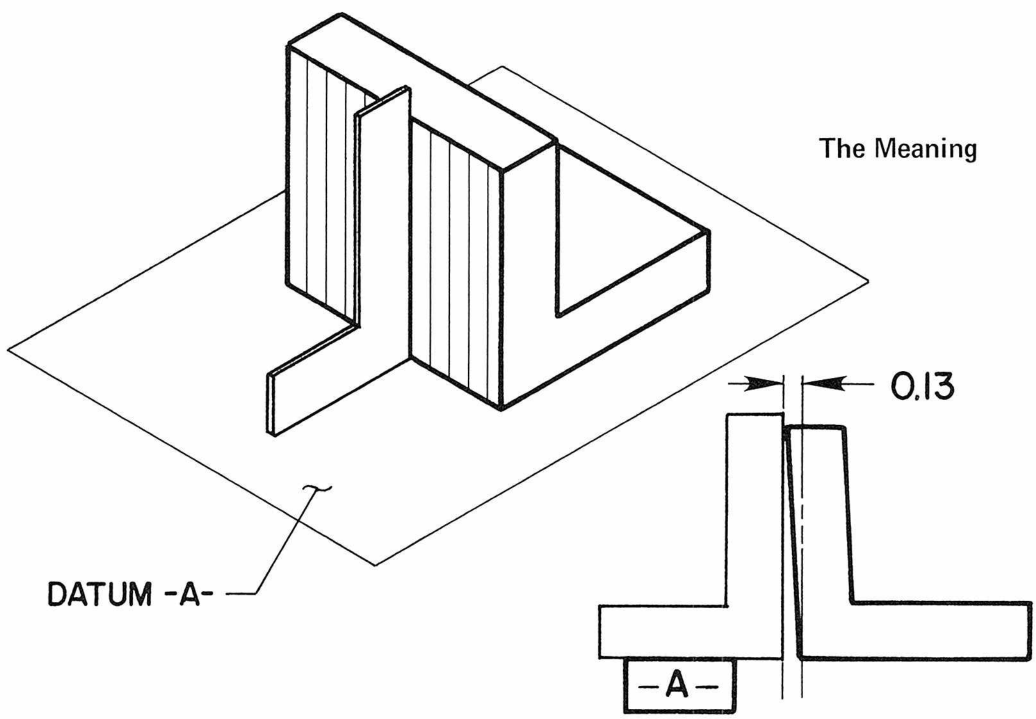
Example 5-21

In this situation the tolerance zone is specified as having a diameter descriptor. So, the center of the object must be within a cylindrical zone that is 90° to the datum.

Another possibility is that rather than having an entire surface of an object perpendicular to a datum, maybe it would be preferred to have any single line of the surface perpendicular. Example 5-22 demonstrates this possibility.



The Drawing



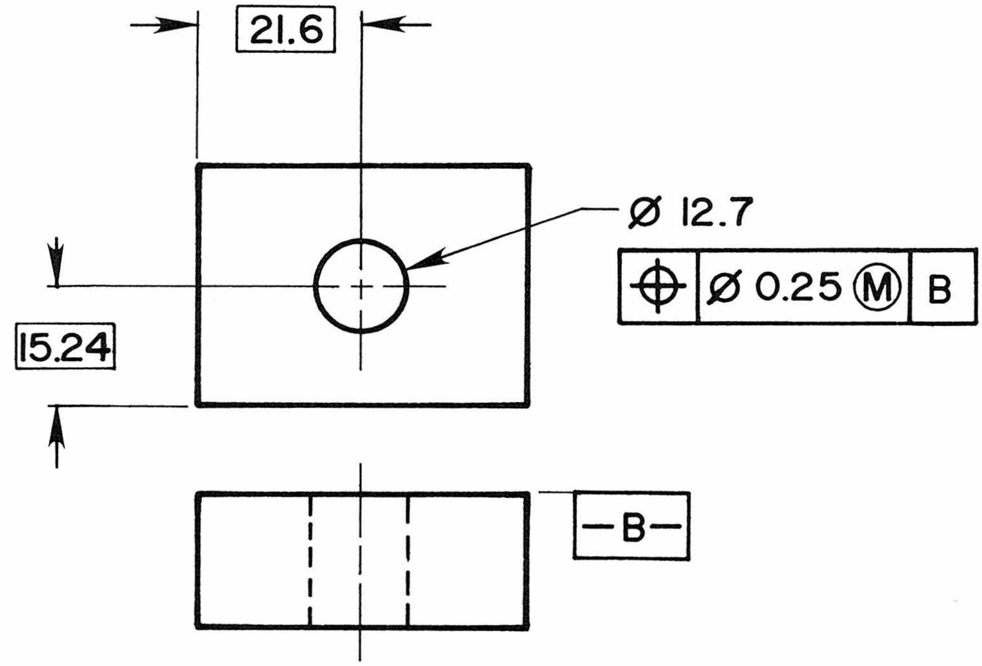
The Meaning

Example 5-22

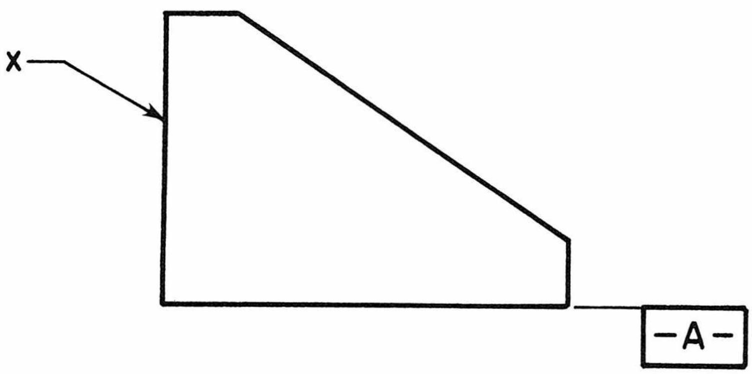
When any single line element of the object must be perpendicular, then the terms "each element" must be indicated below the feature control frame.

Try these:

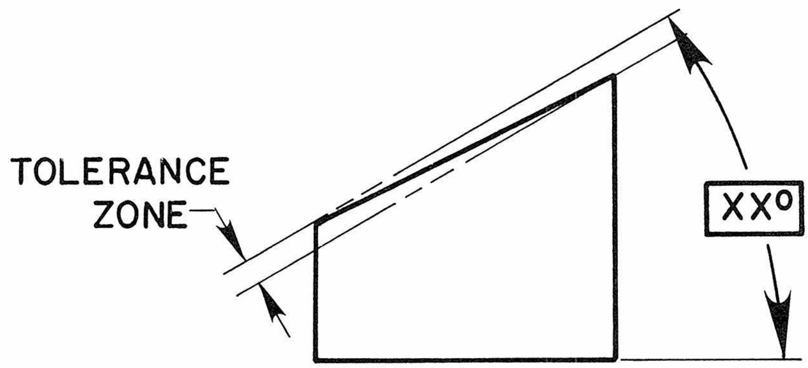
Provide a perpendicularity tolerance so that the 12.7 diameter hole below will be perpendicular to datum -B- by 0.076.



Dimension the object below so that each element of the surface marked "x" will be perpendicular to datum -A- by 0.13.



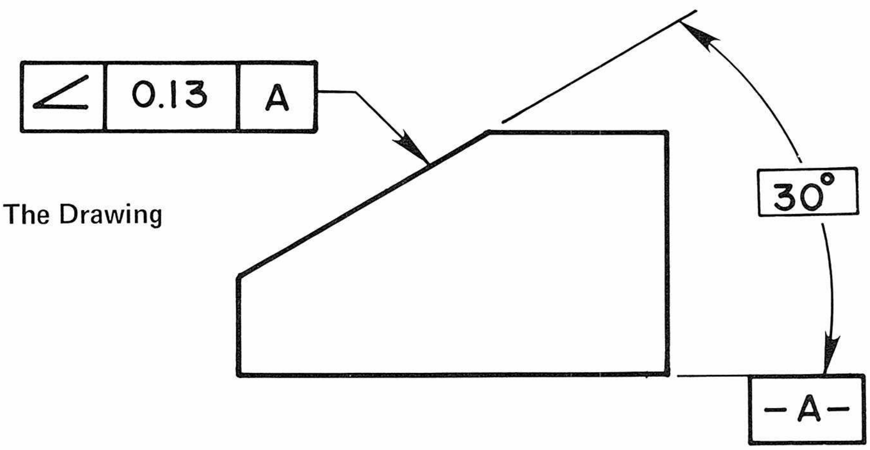
∠ ANGULARITY



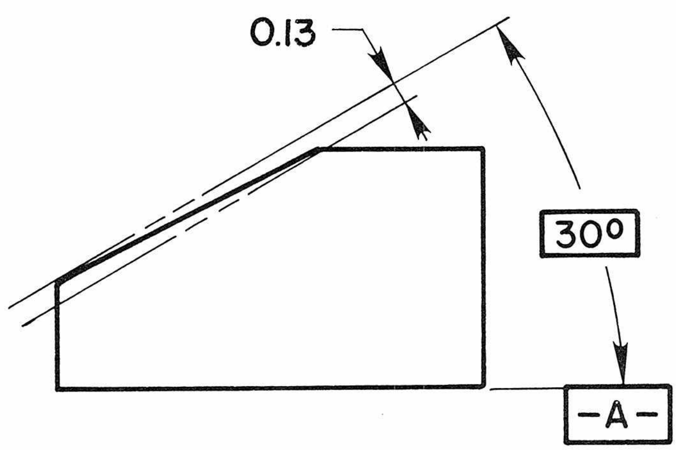
Example 5-23

Example 5-23 shows a typical situation of what happens to an object when an angularity tolerance is applied. The angle must be identified with a basic dimension. Study Example 5-24 to determine how this is done on a drawing. The feature control frame may also be attached to the extension line that projects from the inclined surface.

The Drawing



The Meaning

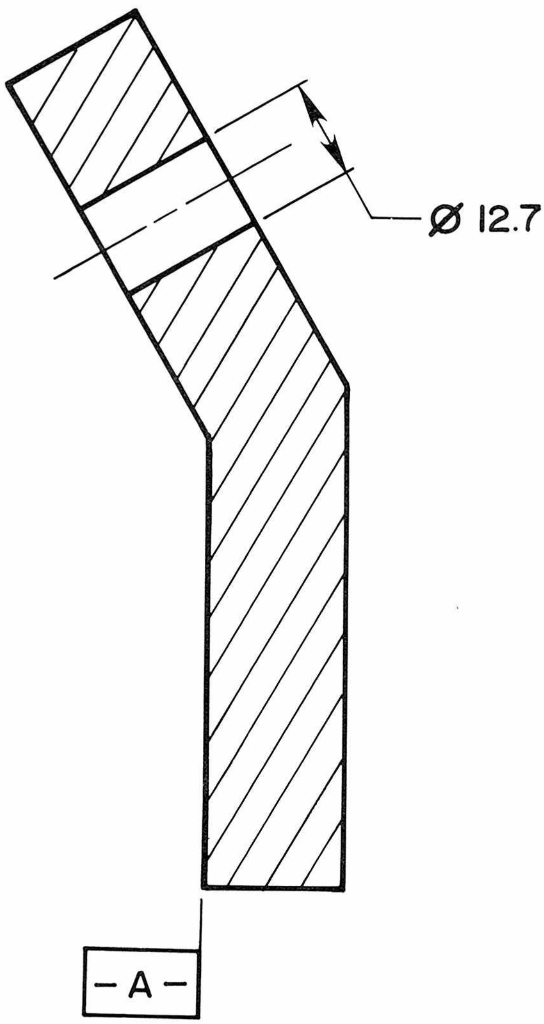


Example 5-24

The axis of a hole or a cylindrical object can also be dimensioned with angularity if the object is at an angle to the datum. The method is similar to that used in perpendicularity except that the angle must be a basic dimension and the geometric characteristic symbol is angularity.

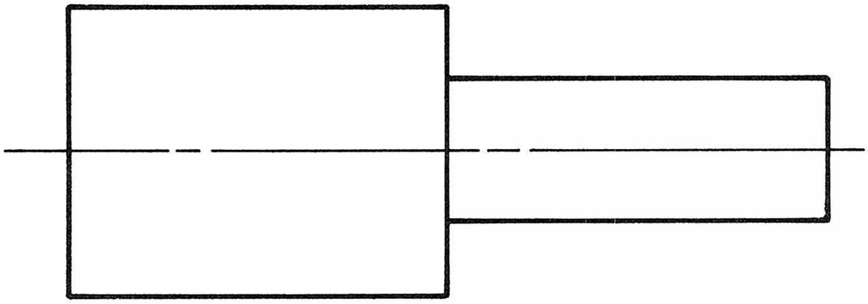
Try this:

The 12.7 diameter hole below is at a 60° relationship to datum -A-. Provide a feature control frame that will insure an angularity tolerance of 0.13.



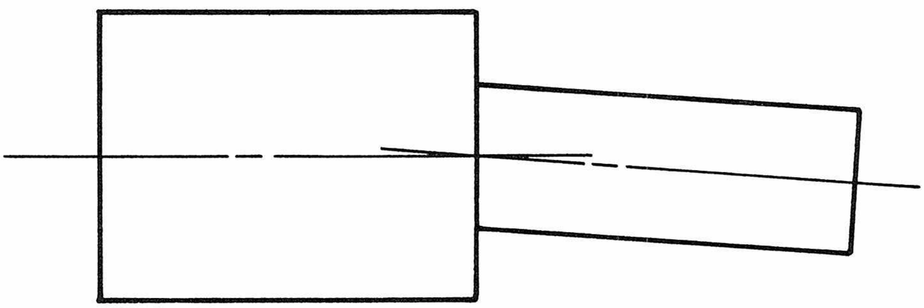
CONCENTRICITY

Concentricity is used to establish a relationship between the axes of two or more cylindrical parts of an object. Perfect concentricity exists when the axis of each cylindrical part falls on the same line. Example 5-25 shows perfect concentricity.



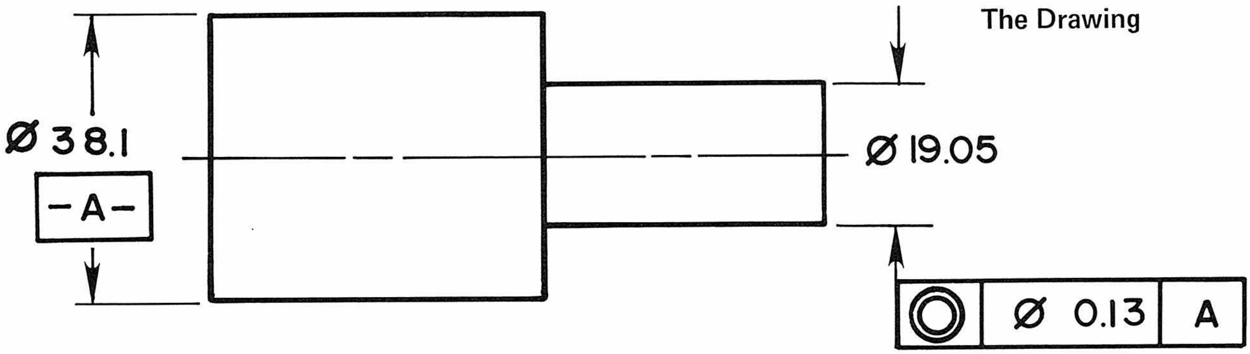
Example 5-25

Example 5-26 shows what happens when concentricity is not perfect.



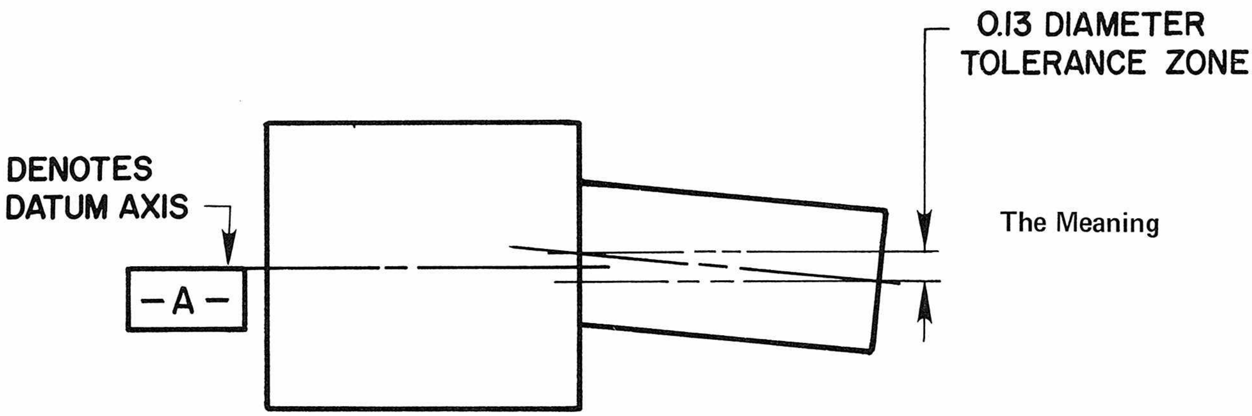
Example 5-26

Concentricity can be dimensioned on an object as demonstrated in Example 5-27. The feature control frame may also be adjacent to the 19.05 diameter dimension.



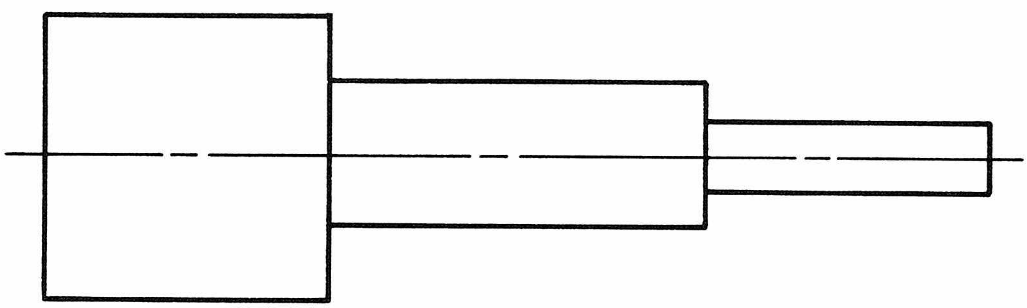
Example 5-27

The zone descriptor for concentricity is always diameter.



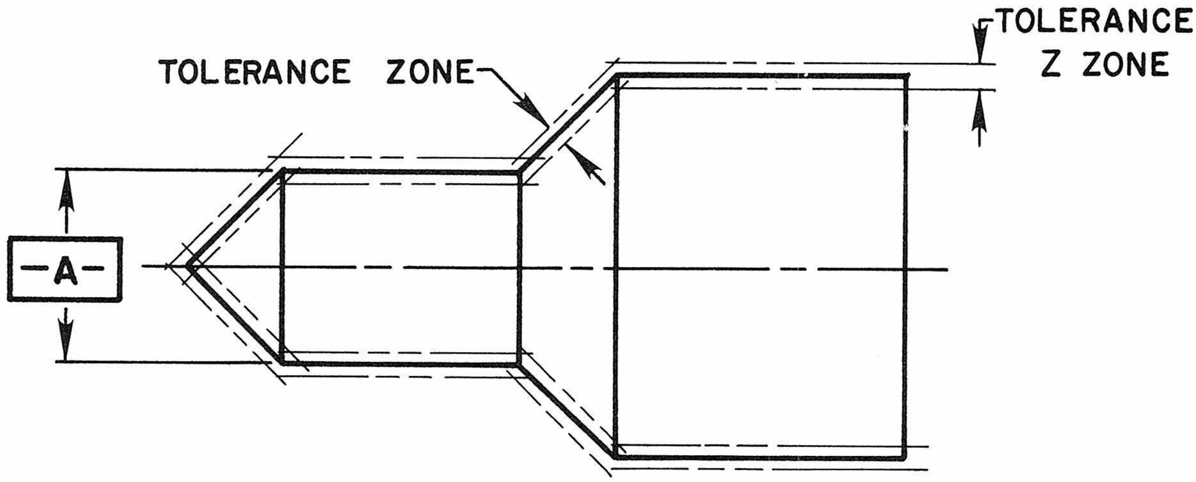
Try this:

The large diameter of the object below will be datum -A-. The middle diameter and the small diameter will be concentric to datum -A- by 0.13. Dimension completely!



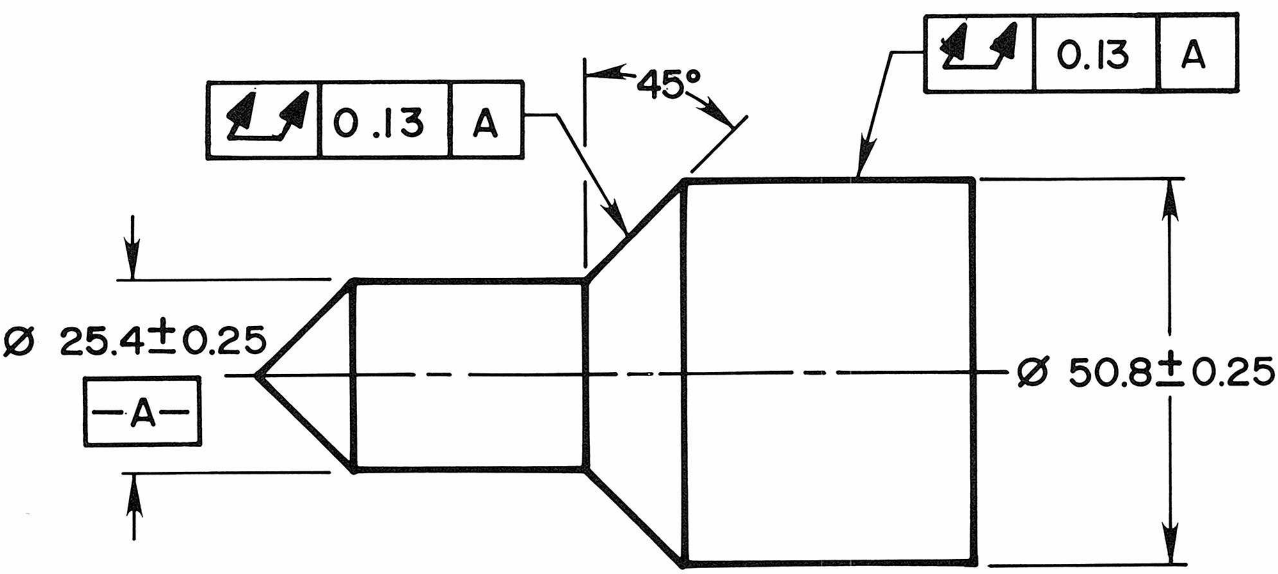
RUNOUT

Runout controls form and the interrelationship to a datum axis. See Example 5-31.



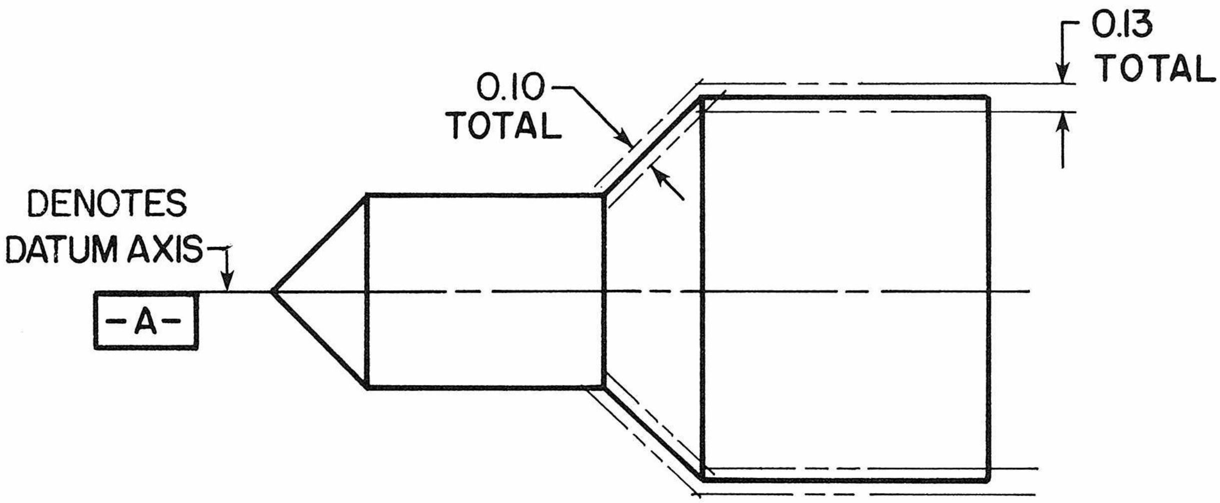
Example 5-31

There are two types of runout. Total runout ( ) provides a control of all elements on the entire surface. Example 5-32 shows total runout.

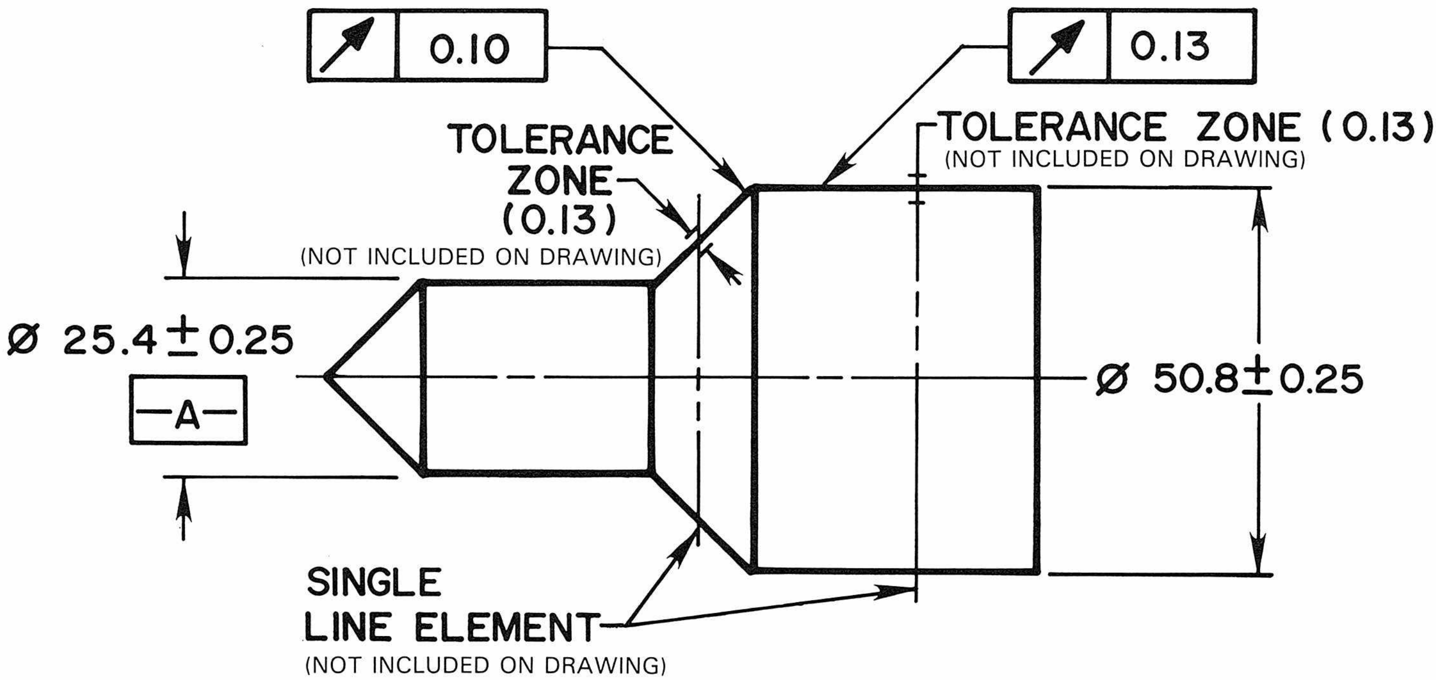


Example 5-32

Example 5-32 as a drawing means this:



The other form of runout (↗) is the control of any single circular element of the feature. Example 5-33 shows the application and the meaning of this method of runout.

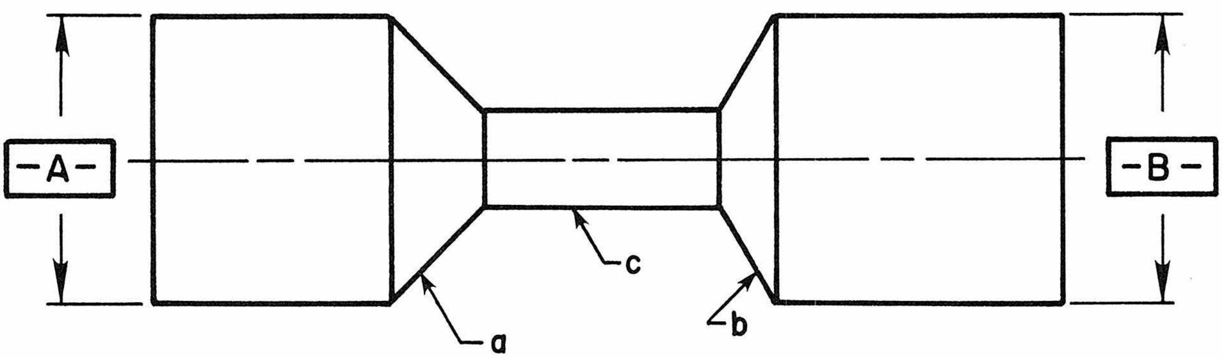


Example 5-33

The runout of a part feature is checked by rotating the part 360°. The resultant total indicator reading must be within the tolerance zone.

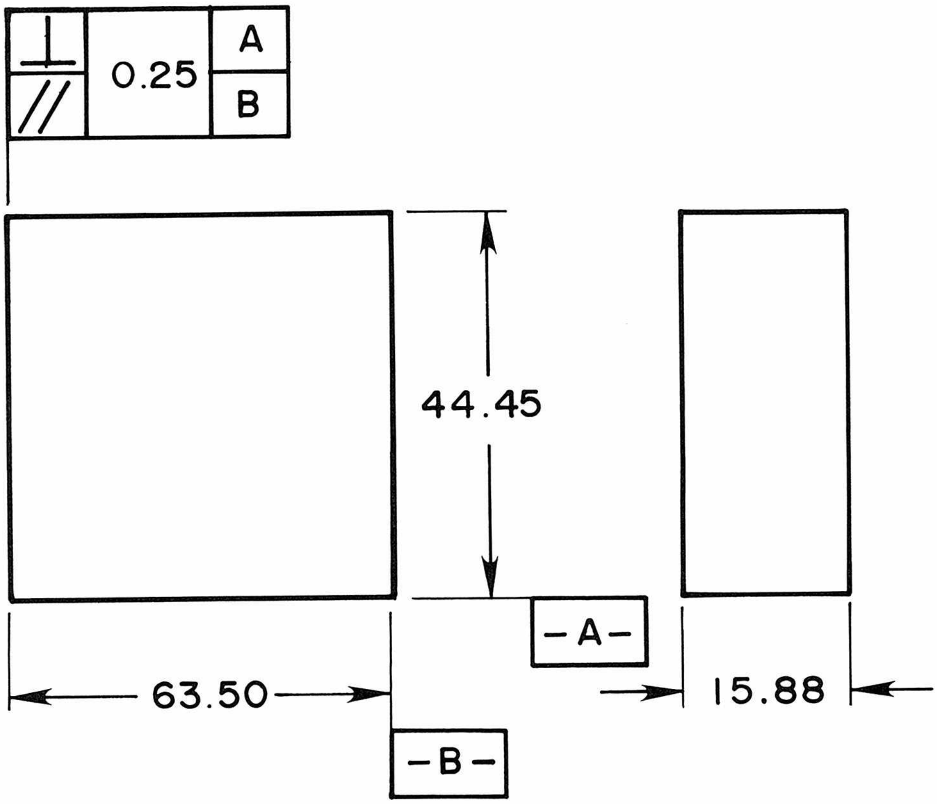
Try this:

Dimension the object below so that the surface ''a'' will have a total runout of 0.076 to datum -A-. Allow a 0.076 single line circular runout tolerance to datum -B- at surface ''b.'' Also, provide a 0.05 total runout tolerance to surface ''c.''



COMBINATION OF GEOMETRIC CHARACTERISTICS

Often times it may be necessary to combine geometric characteristics to achieve certain design considerations. Example 5-34 shows a typical situation where the surface in question must not only be perpendicular to datum -A-, but also parallel to datum -B-.



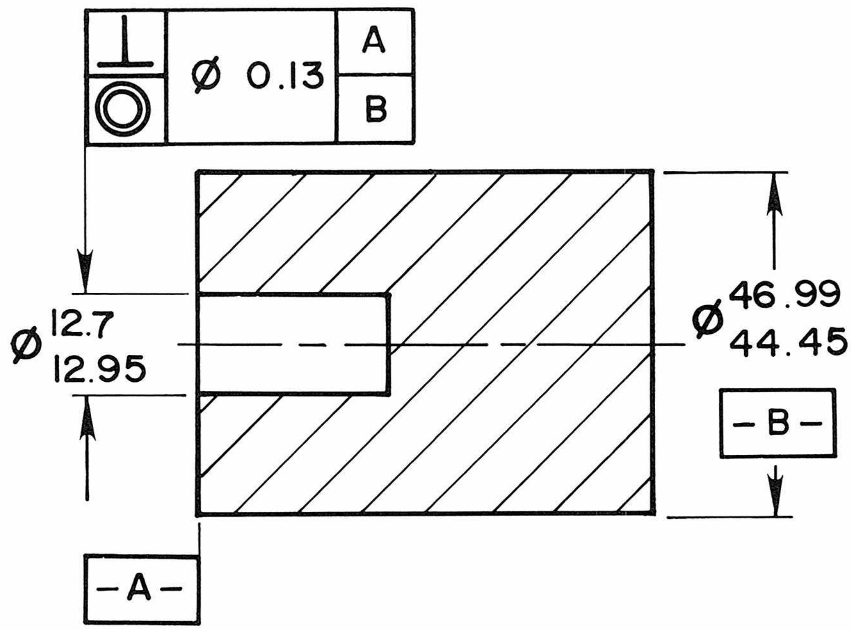
Example 5-34

This allows the engineer the versatility of providing uniform perpendicularity while holding parallelism.

Different tolerance zones could be indicated by using a feature control frame that looks like this: (Same tolerance zone may also be done in this manner.)

⊥	0.25	A
//	0.13	B

Example 5-35 shows another situation that lends itself to a combination of geometric characteristics.



Example 5-35

NOTE: For additional practice in handling geometric tolerances in controlling feature variation, see Problem 2, page 124.

GEOMETRIC DIMENSIONING AND TOLERANCING

Post Test 5 Name:____________________________

Note: Draw all problem drawings full scale. Use all proper drafting techniques. Use proper size symbolism. Do not sketch.

1. Show an example of an object properly dimensioned with:

(A) Straightness Tolerance.

(B) Flatness Tolerance.

Post Test 5

(C) Circularity Tolerance.

(D) Cylindricity Tolerance.

(E) Parallelism Tolerance.

Post Test 5

(F) Perpendicularity Tolerance.

(G) Angularity Tolerance.

(H) Concentricity Tolerance.

Post Test 5

(I) Surface Profile Tolerance.

(J) Combination of perpendicularity and parallelism on the same object.
Use different tolerance zones.

Post Test 5

2. Circle the "T" if the statement is true. Circle the "F" if the statement is false. Reword all false statements so that they will be true. Read each statement carefully.

T F (A) Unit straightness may be used if the part must be controlled per unit of measure as well as over the total length.

T F (B) Specific area flatness should be avoided on very large parts.

T F (C) The zone description of a circularity tolerance is diameter.

T F (D) Cylindricity is identified by an on radius tolerance zone that establishes two perfectly concentric cylinders.

T F (E) The profile of a line must be established between two given points of an object.

T F (F) A parallelism tolerance zone must be between the size tolerance of an object.

T F (G) The perpendicularity of a shaft or hole to another surface establishes a diameter tolerance zone.

T F (H) The terms "each element" must be applied to a perpendicularity feature control symbol.

T F (I) Angularity must have a basic angular relationship to a datum.

T F (J) The zone descriptor of a concentricity tolerance is always "on R."

T F (K) Concentricity is used to establish a relationship between the axis of two or more cylindrical features of an object.

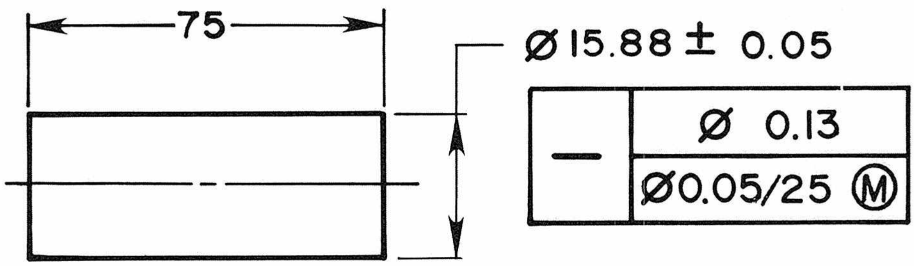
T F (L) The profile tolerance zone may be bilateral or unilateral.

T F (M) Surface straightness may violate perfect form at MMC.

T F (N) Geometric tolerances and related references imply RFS unless other material condition symbols are specified (except: position).

Post Test 5

3. Fill in the appropriate straightness values permitted at the produced sizes shown. (Axis Control Review Chapter 4)



Actual Feature Size	Per Inch	Max. Acceptable Over Total Length
15.93		
15.90		
15.88		
15.875		
15.85		
15.83		

Chapter 6

POSITION

This chapter singles out concepts of position for the purpose of locating features from a datum or from related features. Keep in mind, however, that certain geometric characteristics that were discussed in Chapter 5 also control position of features to a certain extent. Concentricity and runout are specific types of specifications that control location of one or more features to another.

The use of position dimensioning concepts provides some of the greatest advantages to mass production in comparison to conventional methods. You will see how the coordinate dimensioning system limited the actual location of features to a rectangular tolerance zone. Using position, this tolerance zone changed to a circle, thus increasing possible location area by about 57 percent. This increases interchangeability of parts without changing the effect of the dimensioning.

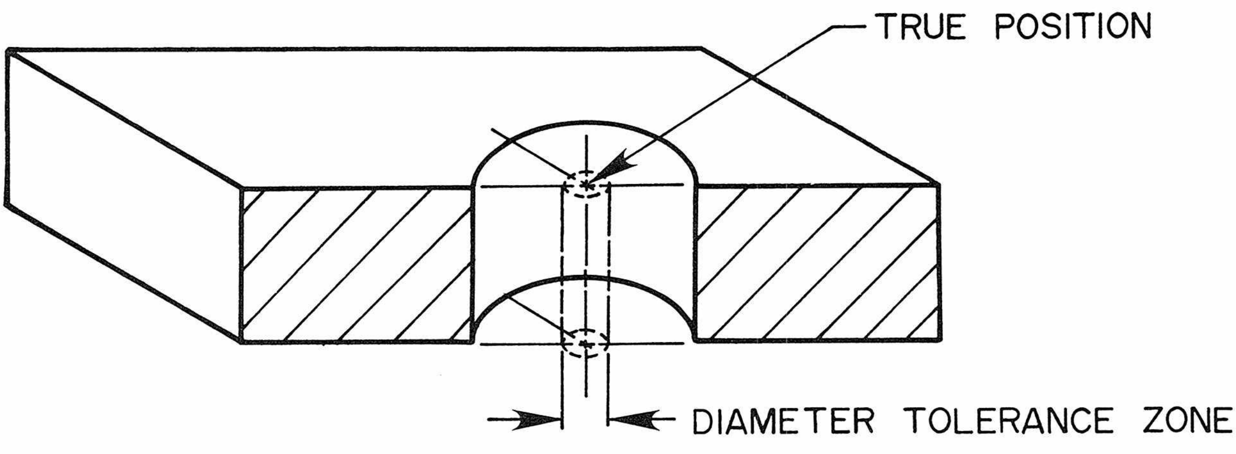
Also, the addition of a Maximum Material Condition modifier to a position callout provides for location of features at MMC. This allows the tolerance zone to increase in size as the size dimension is produced at less (greater, for holes) than MMC. This also allows greater flexibility in the acceptance of mating parts.

⌖ POSITION

Position tolerances must be modified with MMC, LMC, or RFS. MMC is most common.

True position is the theoretically exact location of the center line of a feature.

Position tolerances control the location of a cylindrical tolerance zone within which the center line of a feature is located. See Example 6-1.

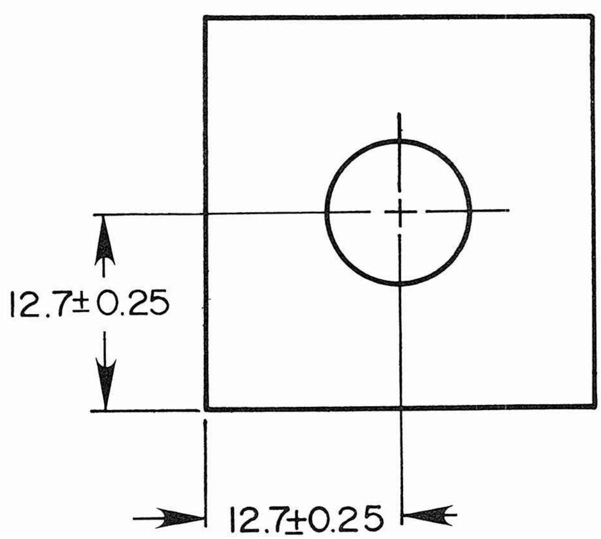


Example 6-1

Position tolerances are established with a diameter tolerance zone descriptor.

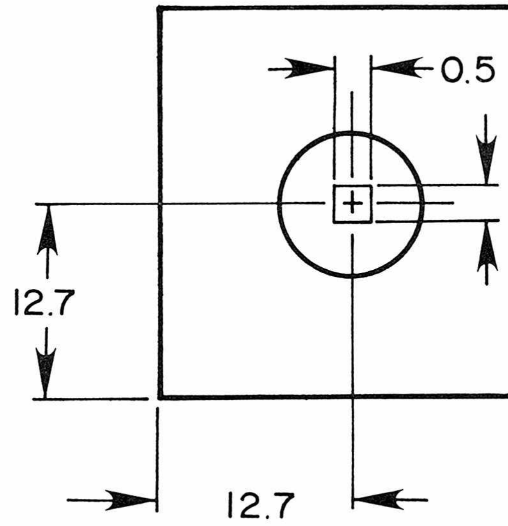
CONVENTIONAL TOLERANCING VS POSITION TOLERANCING

The location of a hole using conventional tolerancing methods is shown in Example 6-2.



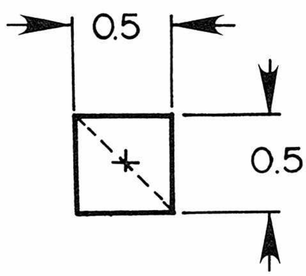
Example 6-2

The 12.7 ± 0.25 locational dimensions establish a total tolerance zone of 0.5. This tolerance zone is square as you can see in Example 6-3.



Example 6-3

Let's look at this tolerance a little closer and analyze what it really means.

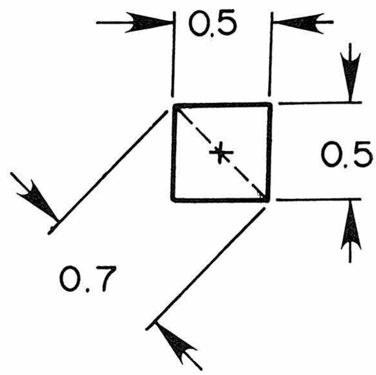


The tolerance zone above means that the center of the hole can fall anywhere within the square area and be an acceptable part.

The application of position tolerancing will allow us to increase the acceptable tolerance zone. Consider this:

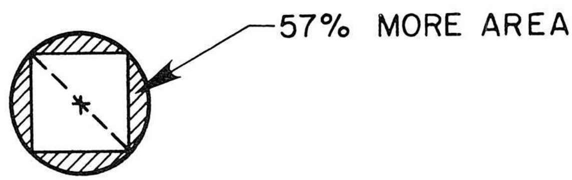
> The diagonal of the square tolerance zone is the greatest distance that allows variation in the location of the center.
>
> The length of this diagonal is equal to a constant of 1.4 times the tolerance of the locational dimensions.

Using the tolerance zone from Example 6-3 will give us 1.4 × 0.5 = 0.7. See Example 6-4.



Example 6-4

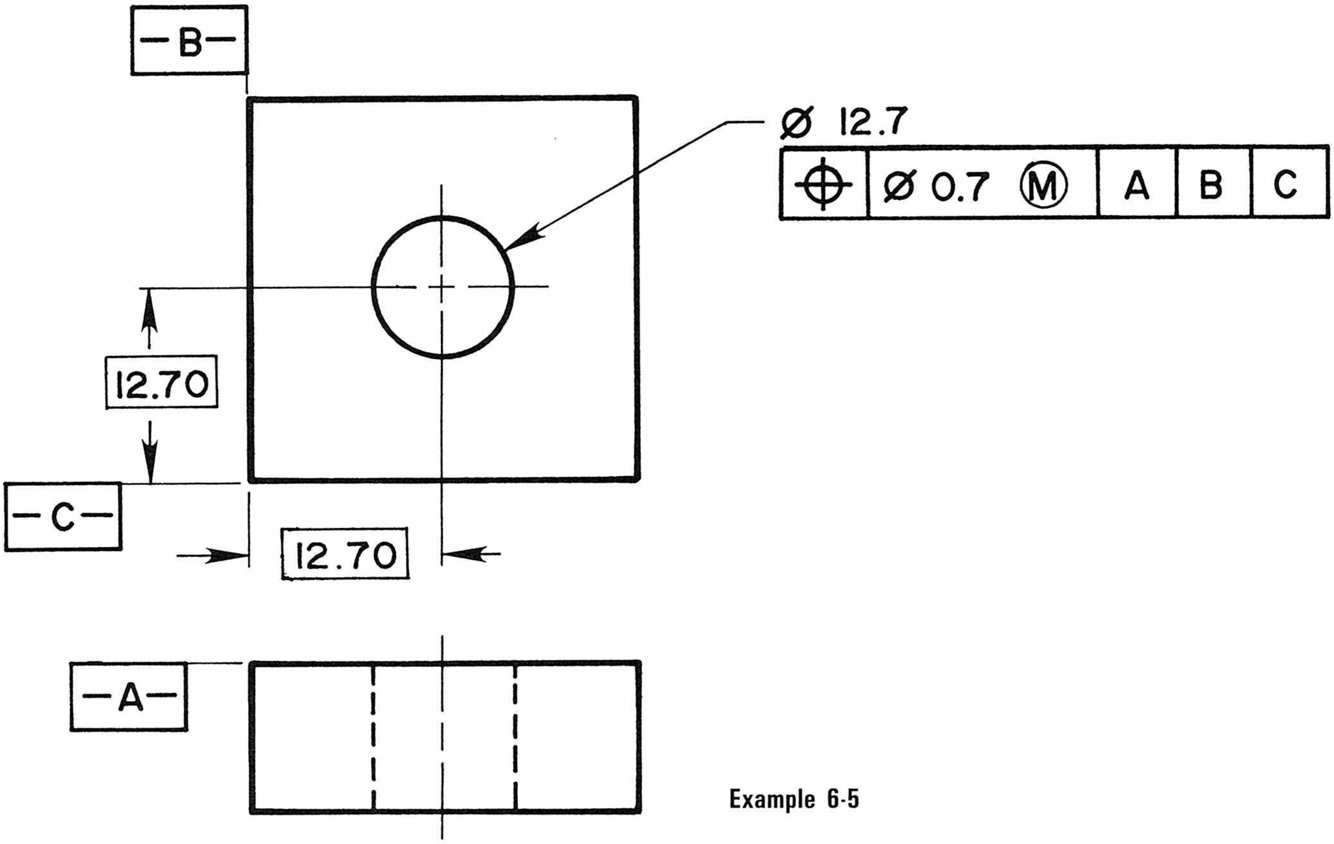
In a position tolerance this 0.7 diagonal will become a diameter tolerance zone that looks like this:



It has been proven that the diagonal tolerance zone is acceptable in any direction, thus creating a circular tolerance zone. The result of this is an increase of 57% of permissable area for the location of the hole. This increased area is shown in the example above.

With the use of this positional tolerance zone there is an increase of acceptability of mating parts and a reduction in manufacturing costs.

Let's look at how a position tolerance is dimensioned on an object. See Example 6-5.

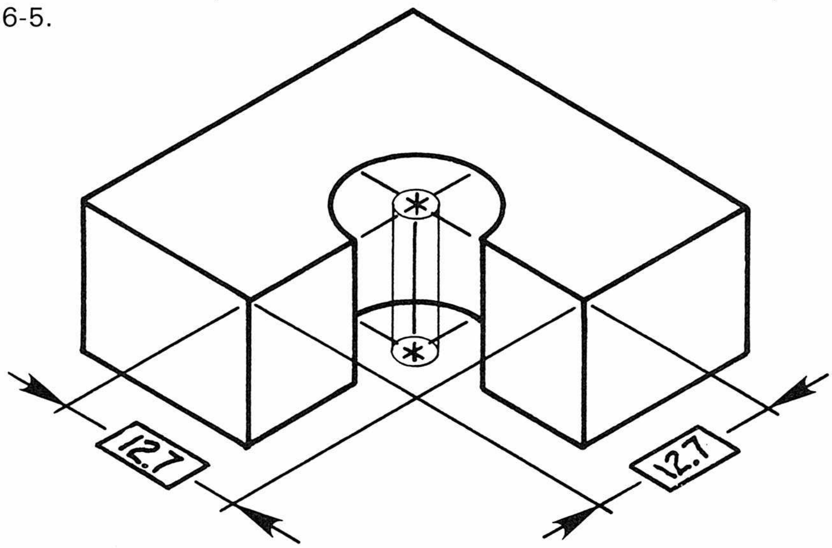


Example 6-5

The hole is located with basic dimensions. An alternative to the basic dimension symbol on all basic dimensions is a general note: UNTOLERANCED DIMENSIONS LOCATING TRUE POSITION ARE BASIC.

Position is usually applied at MMC. However, MMC, LMC, or RFS must always be indicated in the feature control frame adjacent to the tolerance.

Example 6-6 shows the cylindrical tolerance zone that is established by the callout in Example 6-5.



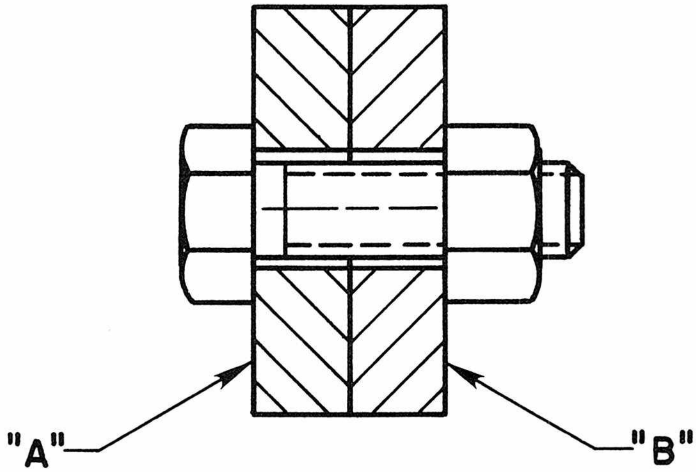
Example 6-6

The center of the hole can be anywhere within the length and diameter of the tolerance zone at maximum material condition.

Observe how position tolerancing can be applied to methods of fastening parts.

FLOATING FASTENERS

Example 6-7 shows what a floating fastener is.



Example 6-7

Parts "A" and "B" in Example 6-7 are fastened together by a bolt and a nut.

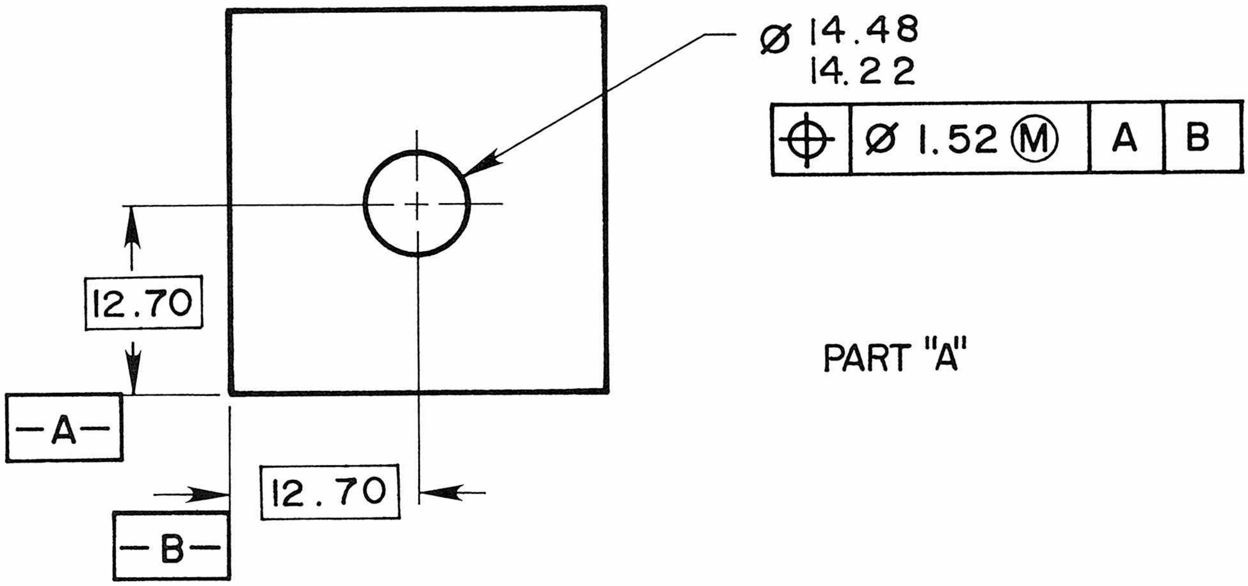
The position tolerance zone is determined this way:

$$\frac{\begin{array}{c}\text{MMC Hole}\\ -\text{MMC Bolt}\end{array}}{\begin{array}{c}\text{Position Tolerance in part ''A'' and part ''B''}\\ \text{(Each part is calculated separately.)}\end{array}}$$

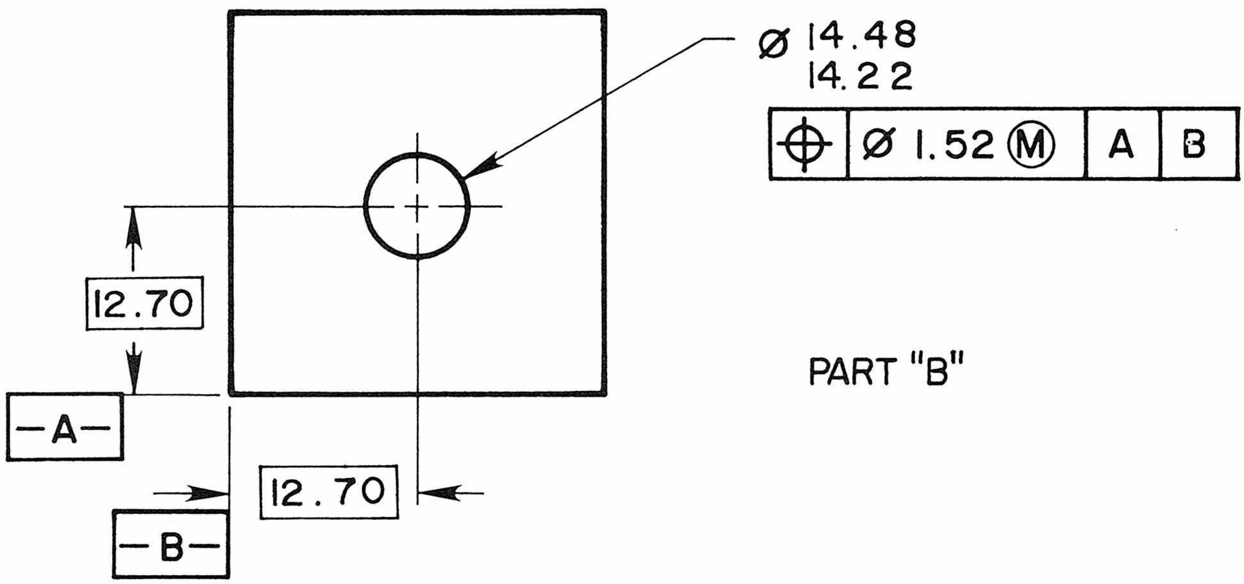
The tolerance zone may be adjusted on each mating part. For instance, the tolerances in the example above could be 1.016 for one part and 2.032 for the other.

In Example 6-8 you can see how the two parts of the object in Example 6-7 are properly dimensioned for a .500 (12.7 mm) diameter bolt. The MMC of the fastener is the nominal size.

MMC Hole		14.22
−MMC Bolt	.500 =	12.70
POSITION		1.52



PART "A"



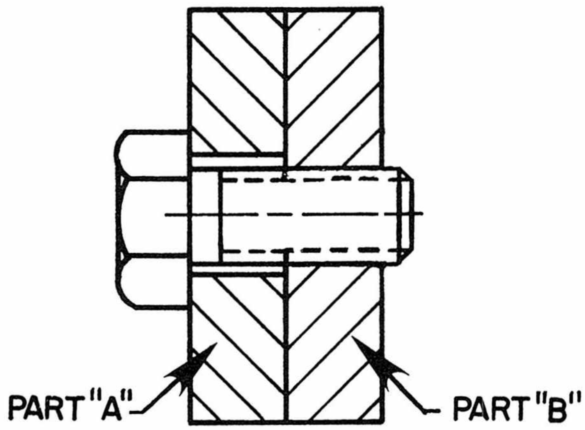
PART "B"

Example 6-8

The drawings for part "A" and part "B" in Example 6-8 show that the tolerance zone is the same for each part, in this situation. In actual practice when both parts are the same only one would be drawn with the note 2 required specified.

FIXED FASTENERS

A fixed fastener is demonstrated in Example 6-9.



Example 6-9

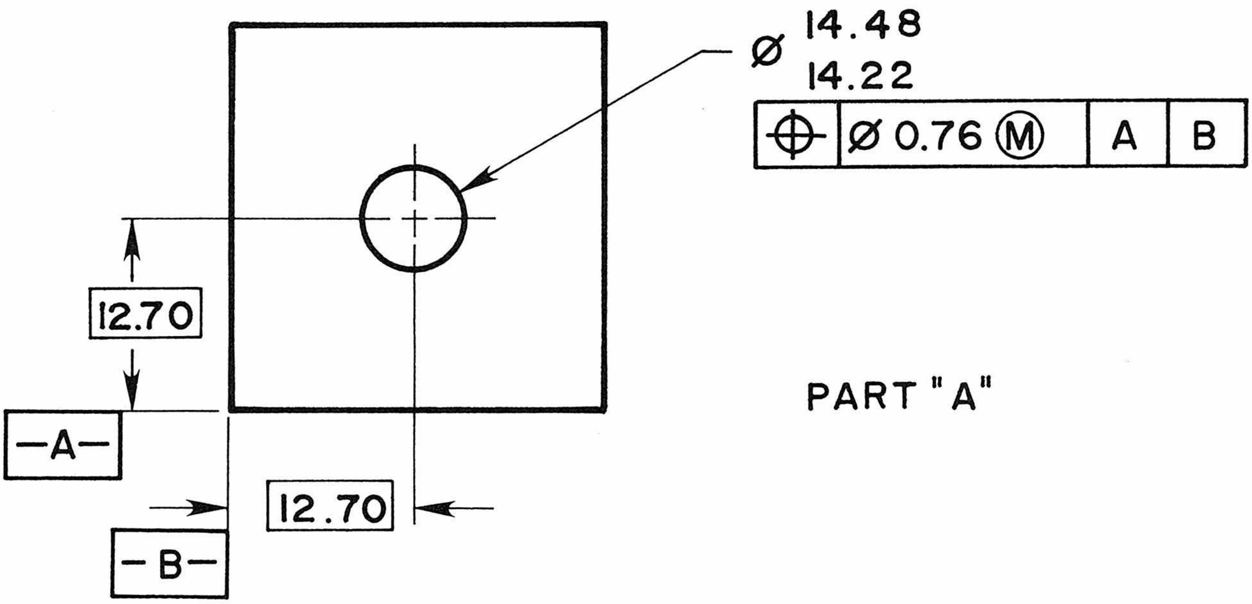
Part "A" has a hole through it and part "B" is threaded so that part "B" becomes a fastening device. Part "A" can be bolted to part "B." The fixed fastener does not require a nut for fastening as does a floating fastener.

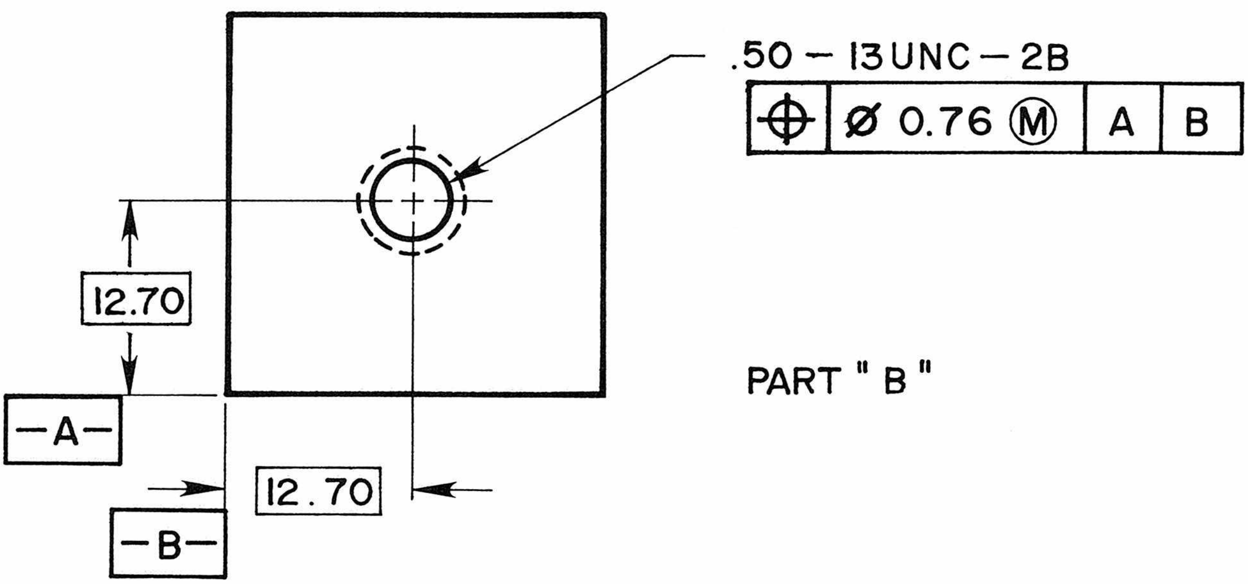
The position tolerance zone is determined this way:

$$\frac{\text{MMC Hole} - \text{MMC Bolt}}{2} = \text{POSITION TOLERANCE FOR EACH PART}$$

See Example 6-10 how the two parts of the object in Example 6-9 are properly dimensioned for a .500 (12.7 mm) diameter bolt.

$$\begin{array}{lr} \text{MMC Hole} & 14.22 \\ -\text{MMC Bolt } .500 = & 12.70 \\ \hline & \dfrac{1.52}{2} = 0.76 \text{ Position Tolerance} \end{array}$$





Example 6-10

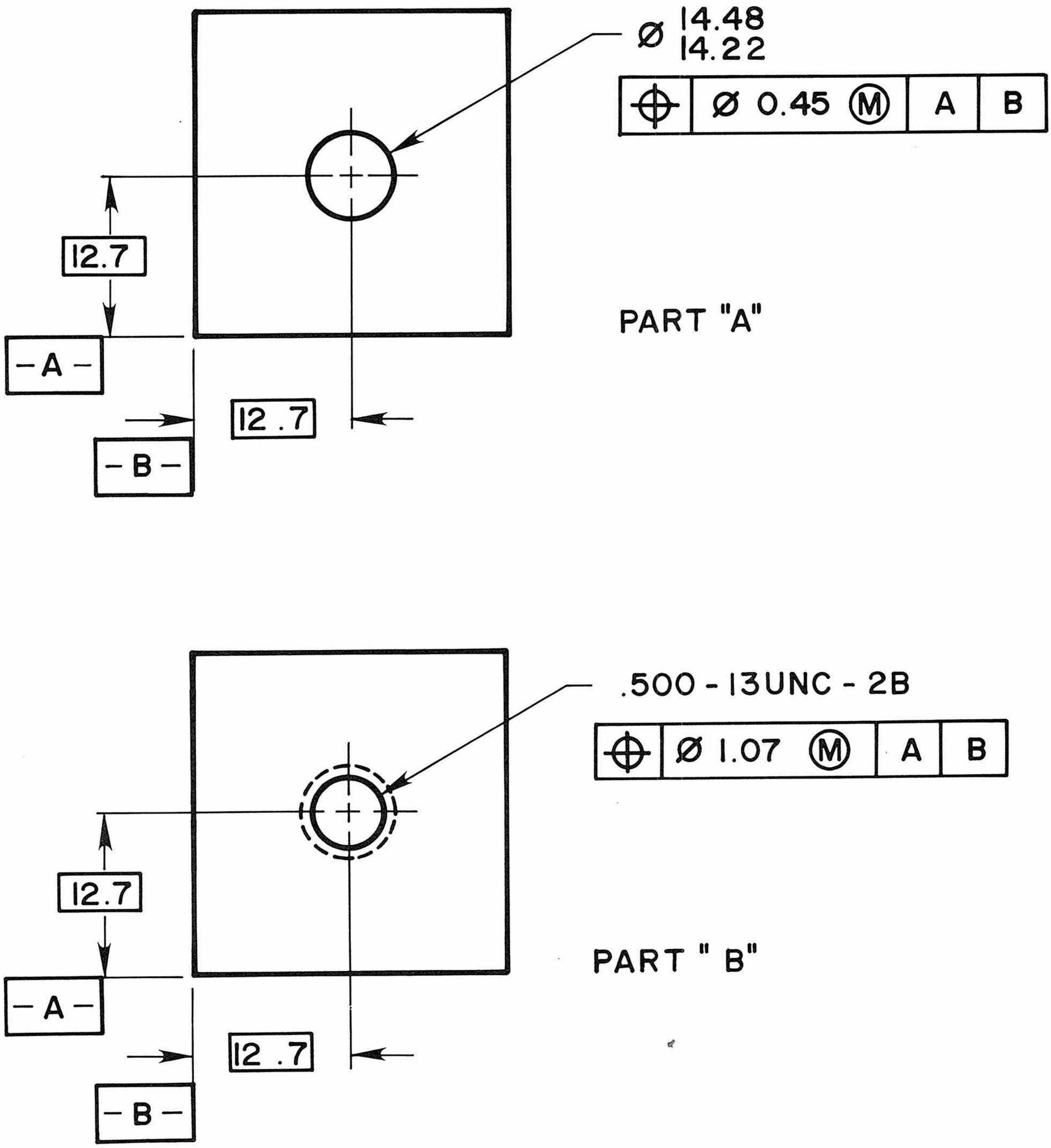
Sometimes it could be possible to have a greater amount of tolerance for the threaded part than for the unthreaded part. For example, if we apply 70% of the tolerance in Example 6-10 to the threaded part and 30% to the unthreaded part, then this is what we would do:

$$\begin{array}{r} \text{MMC Hole } 14.22 \\ -\underline{\text{MMC Bolt } 12.70} \end{array}$$

$$1.52 \times 70\% = 1.07$$

$$1.52 \times 30\% = 0.45$$

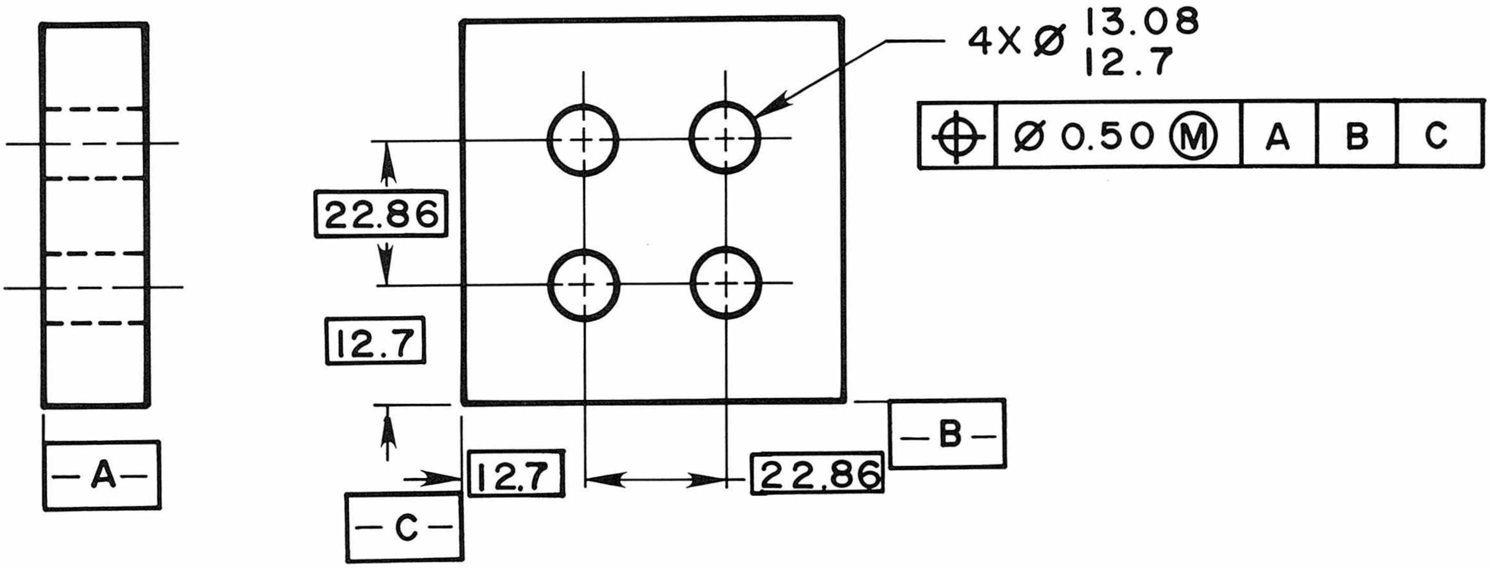
Now, the same object shown in Example 6-10 would be dimensioned as in Example 6-11.



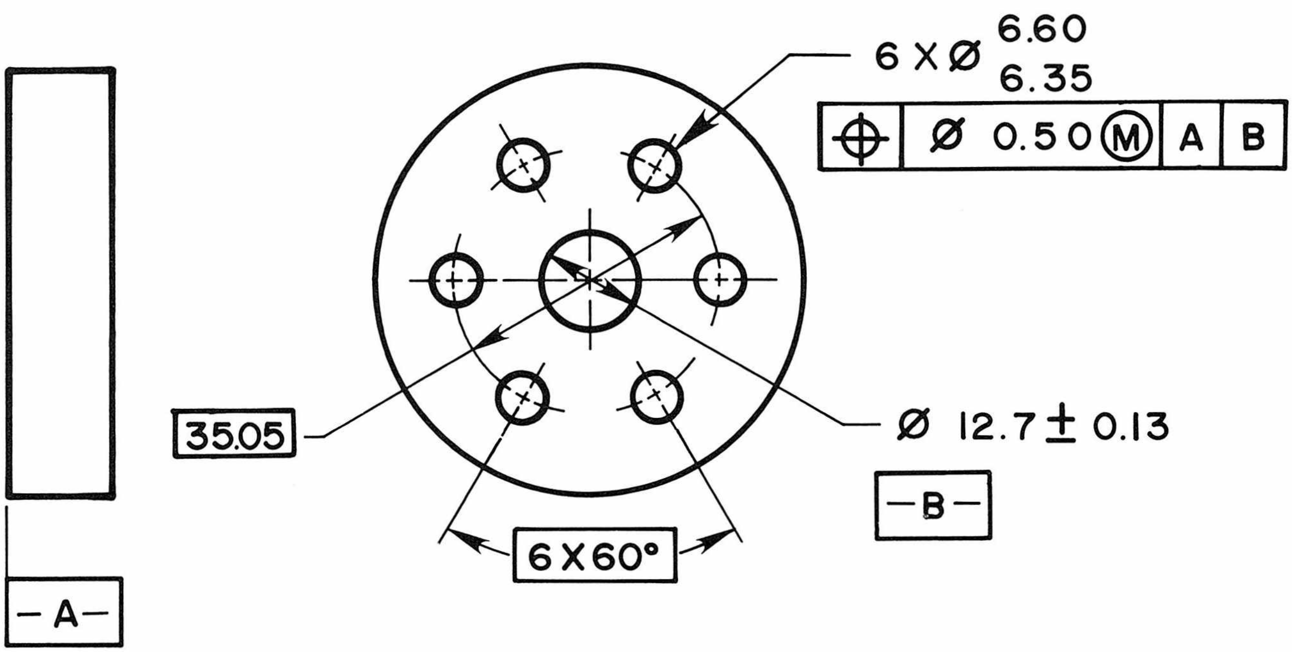
Example 6-11

LOCATING MULTIPLE FEATURES

Multiple features of an object can be dimensioned with positional tolerancing. In this case the location of the features should be dimensioned with basic dimensions from datums and between features. See Examples 6-12 and 6-13 for demonstrations of this.



Example 6-12



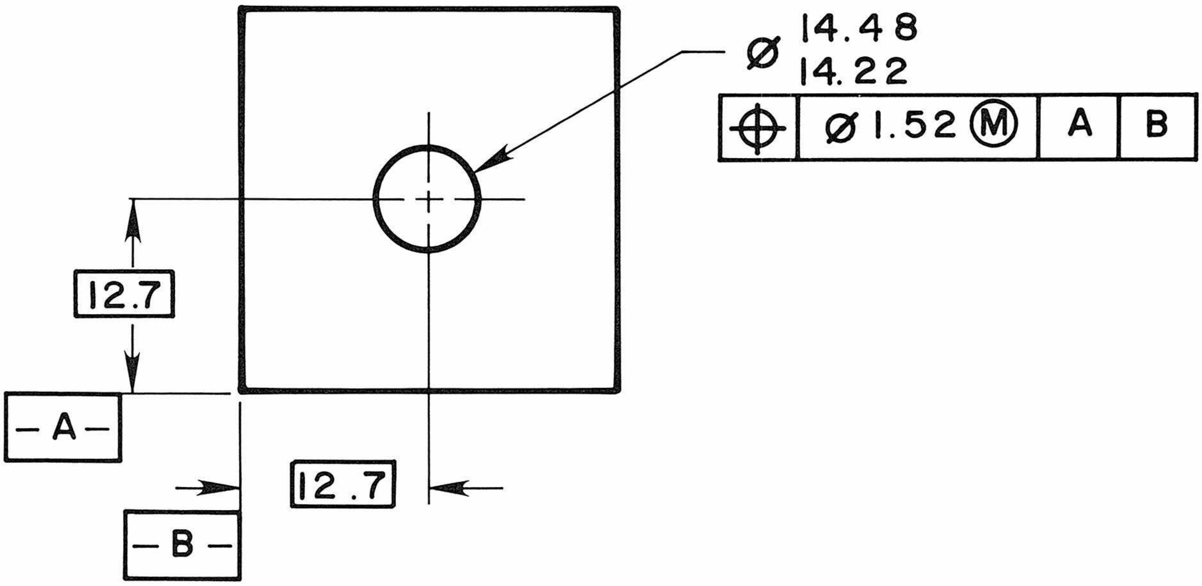
Example 6-13

Note: MMC, LMC, or RFS must be included in the tolerance block.

POSITIONAL TOLERANCE AT MMC

The effect of positional tolerance at MMC is similar to the effect of any geometric tolerance at MMC. The given tolerance only takes effect when the produced size of the feature is at MMC. As the produced size departs from MMC toward Least Material Condition (LMC), then the tolerance zone increases equal to the amount of departure.

Examine the object in Example 6-14 and see what happens at the different produced sizes of the featured hole. Notice how the tolerance increases in diameter as the produced size changes.

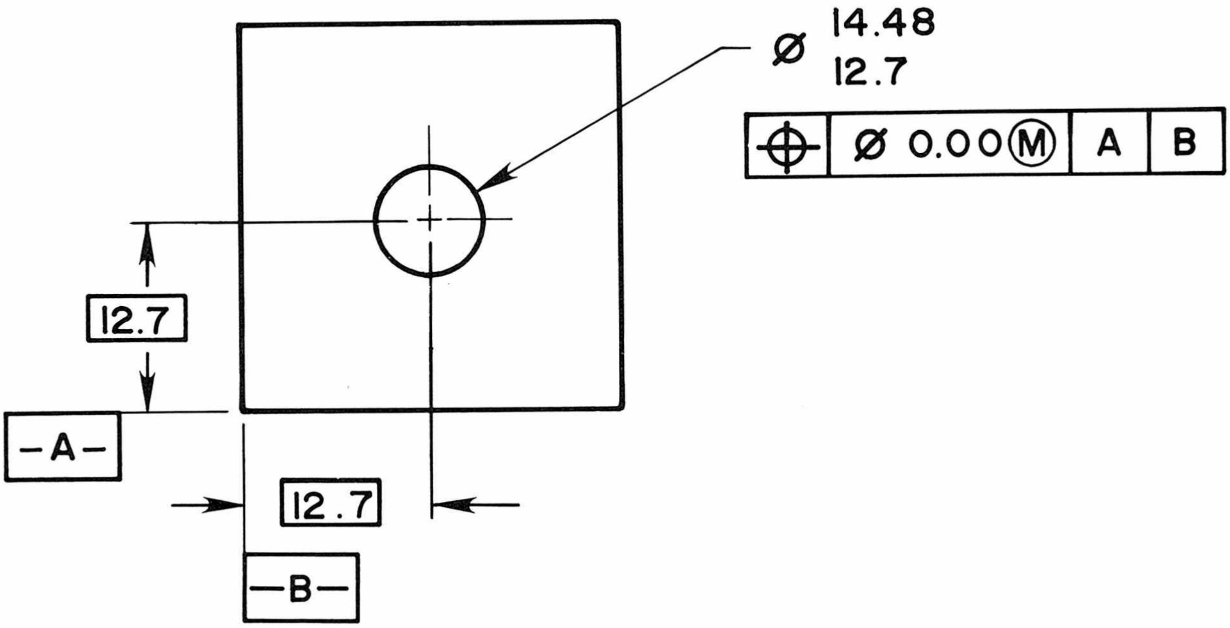


Produced Size	Positional Tolerance
14.22 (MMC)	1.52
14.27	1.57
14.35	1.65
14.48 (LMC)	1.78

Example 6-14

ZERO POSITIONAL TOLERANCE

The consideration of zero tolerance can be applied to any geometric characteristic. Example 6-15 shows zero positional tolerance being used.



Example 6-15

This means:

(A) At MMC the tolerance zone is .000.

(B) The hole is located at perfect true position.

(C) As the actual size departs from MMC the tolerance zone increases equal to the departure. (So, at 13.59 diameter, 13.59 − 12.70 = 0.89 geometric tolerance).

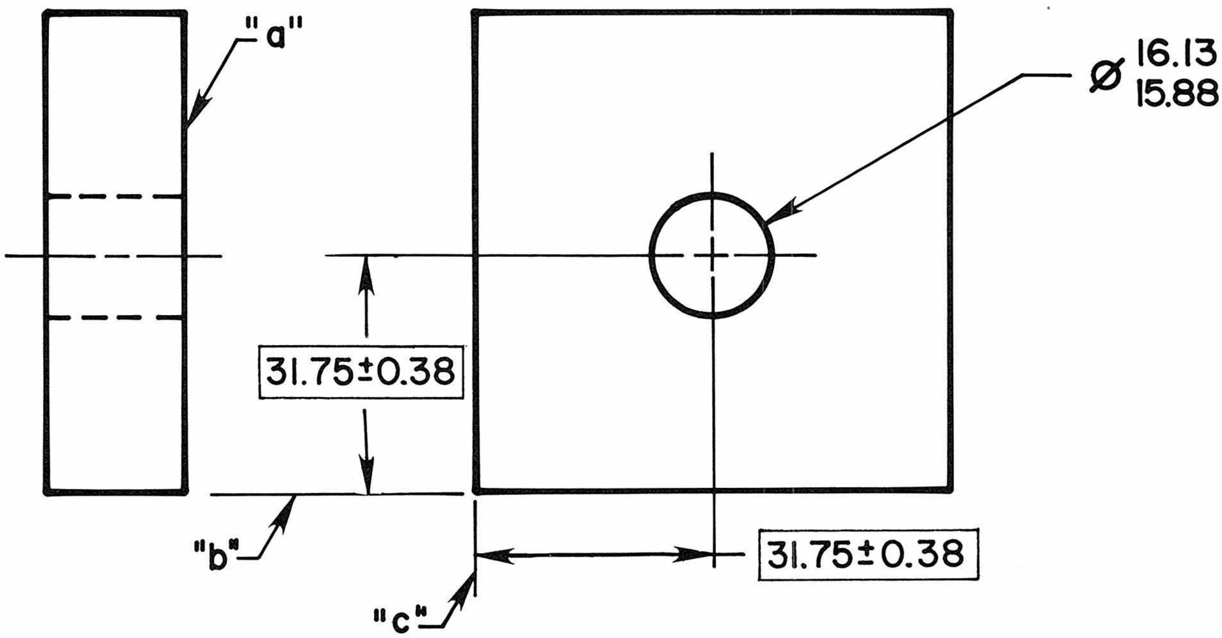
(D) The tolerance is the greatest at LMC. (In Example 6-15, LMC = 14.48 − 12.70 = 1.78 geometric tolerance.)

Produced Size	Positional Tolerance
12.70 (MMC)	0.00
13.20	0.50
13.72	1.02
14.48 (LMC)	1.78

Note: It is suggested that .000 tolerance not be used on threaded holes or studs because the limit of size at the pitch diameter is not that great.

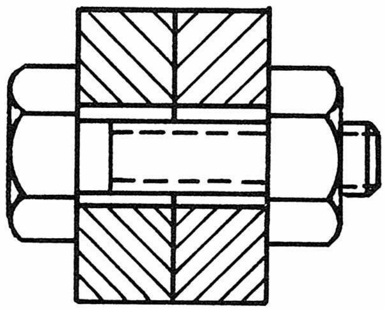
Try these:

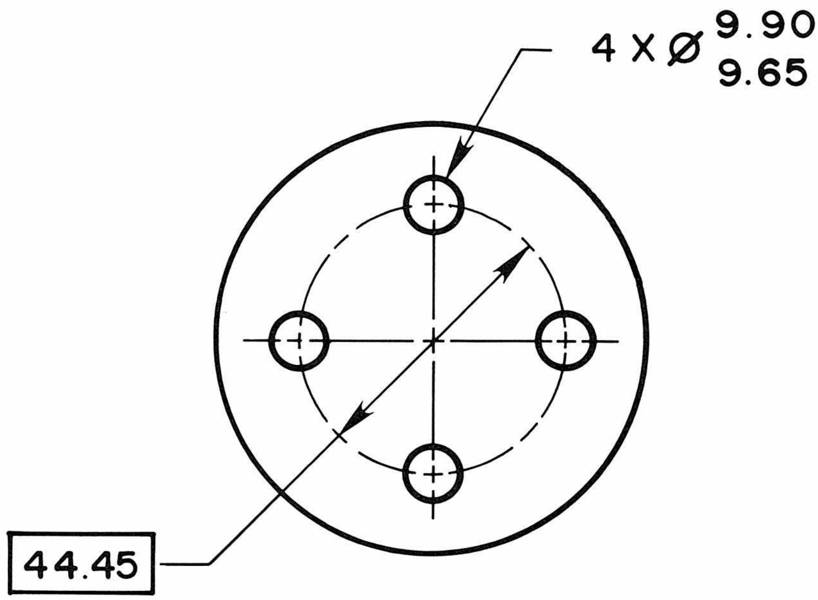
1. Below is an object that is partially dimensioned in a conventional manner. Determine the appropriate positional tolerance zone. Redraw in the space provided and dimension using techniques discussed in this chapter. Note: Datums -A-, -B-, and -C- are marked ''a,'' ''b,'' and ''c'' respectively. (Show your calculations for determining the position tolerance.)



Convert the conventionally toleranced drawing to position tolerance with basic location dimensions and the proper feature control frame in the space above.

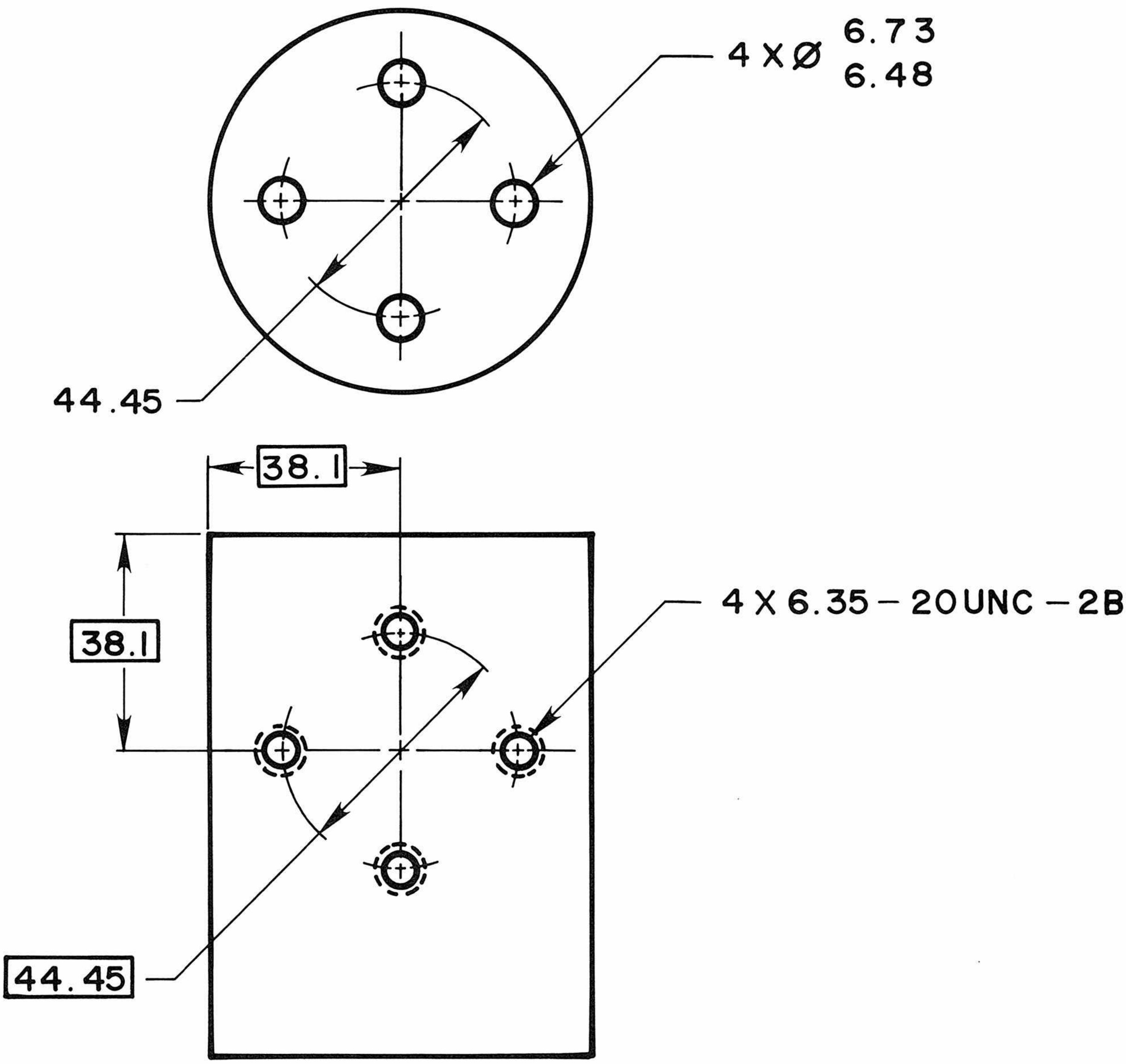
2. Two parts will be fastened together with (4) .375 — 24 UNF machine screws and hex nuts. Calculate the positional tolerance and provide the appropriate feature control frame to the object below. Show all calculations. Dimension <u>completely</u> according to the concepts of positional tolerancing.





Is problem No. 2 a fixed or floating fastener? ______________________

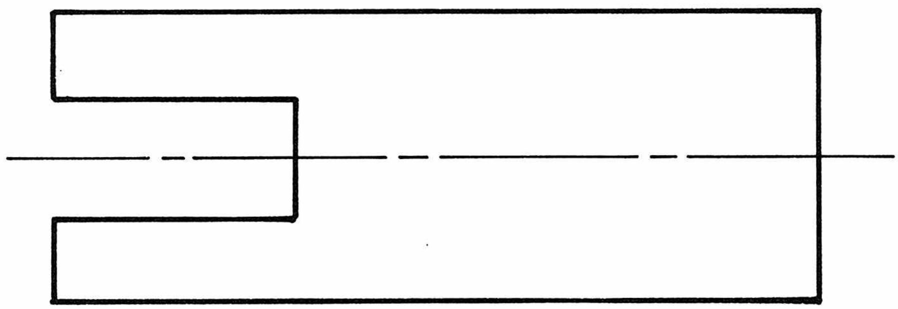
3. The two objects below will be fastened together with (4) .25 – 20 UNC machine screws. Calculate the amount of tolerance required for each part. (Show your calculations.) Dimension each object completely.



Is problem No. 3 a fixed or floating fastener? ______________________

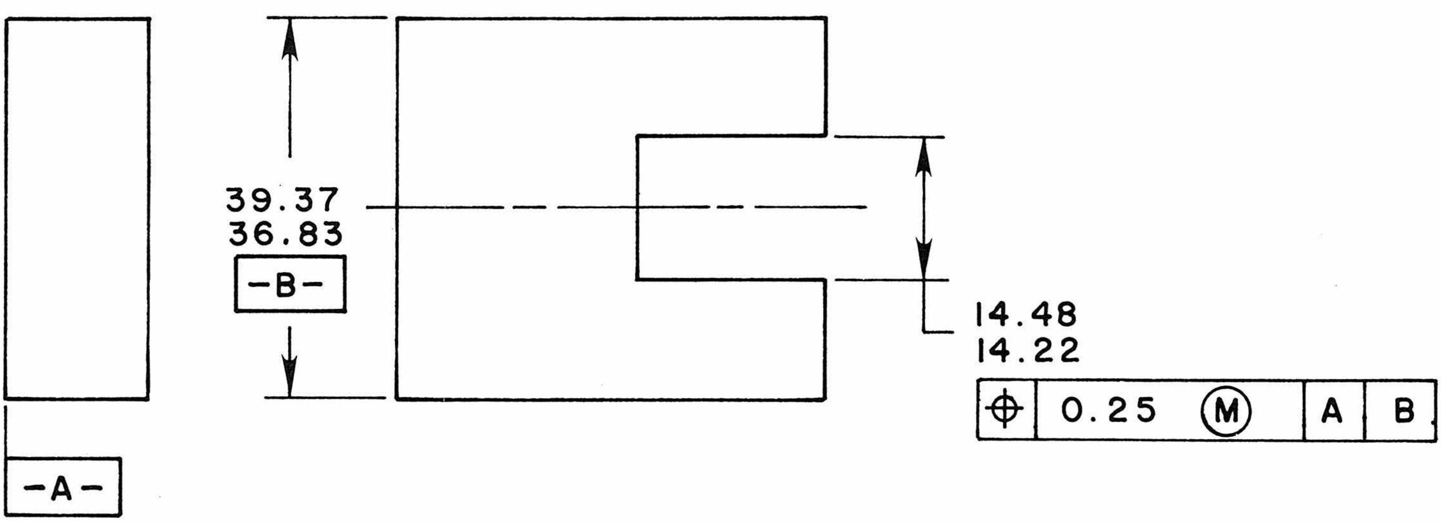
POSITIONAL TOLERANCE LOCATING SYMMETRICAL FEATURES

Symmetry is a centerplane relationship of the features of an object. Just as concentricity, symmetry is perfect when the centerplanes of two related symmetrical features line up as in Example 6-16.



Example 6-16

A positional tolerance is used when it is required to locate a feature symmetrically with respect to the centerplane of a datum feature. The symmetrical relationship may be controlled by a positional tolerance at MMC specification. Look at Example 6-17 for a typical example.



Example 6-17

VIRTUAL CONDITION

Virtual condition is the different size of a feature when considering the combined effect of feature controls and the size. Virtual condition in essence establishes a working zone. Virtual condition takes into consideration the combined effect of feature control and MMC.

Virtual condition is calculated by these methods:

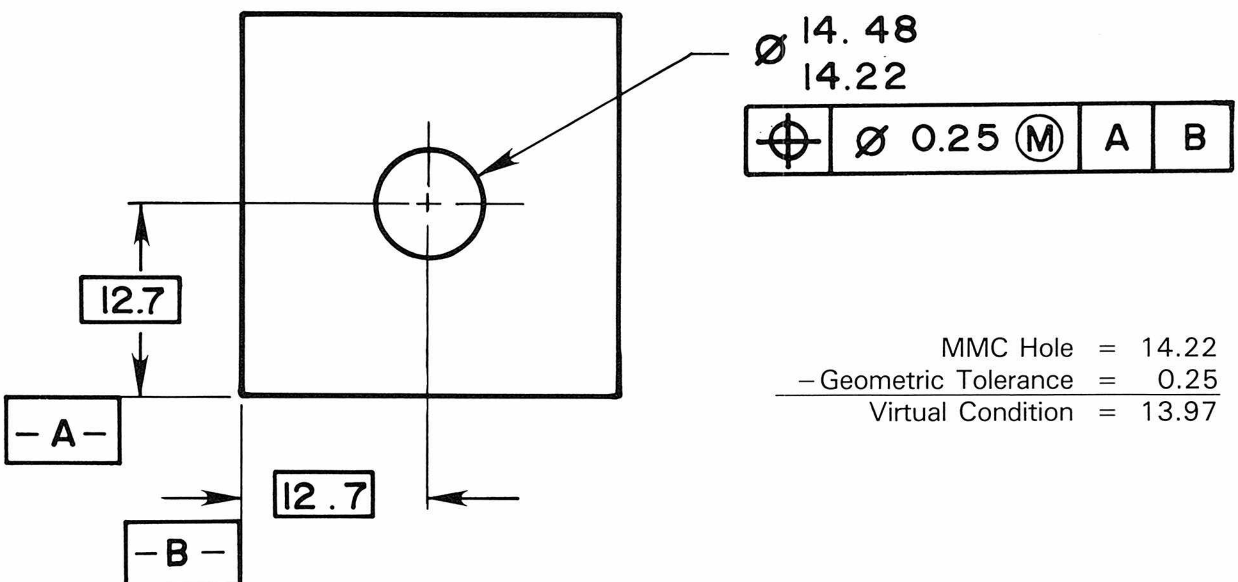
For an external feature (shaft, etc.)

MMC size of feature
+ Related Geometric Tolerance
Virtual Condition

For an internal feature (hole, etc.)

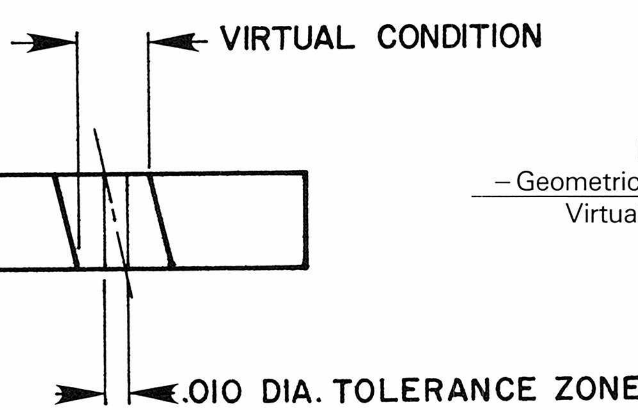
MMC size of feature
− Related Geometric Tolerance
Virtual Condition

Consider Examples 6-18, 6-19 and 6-20 to see how virtual condition is calculated.



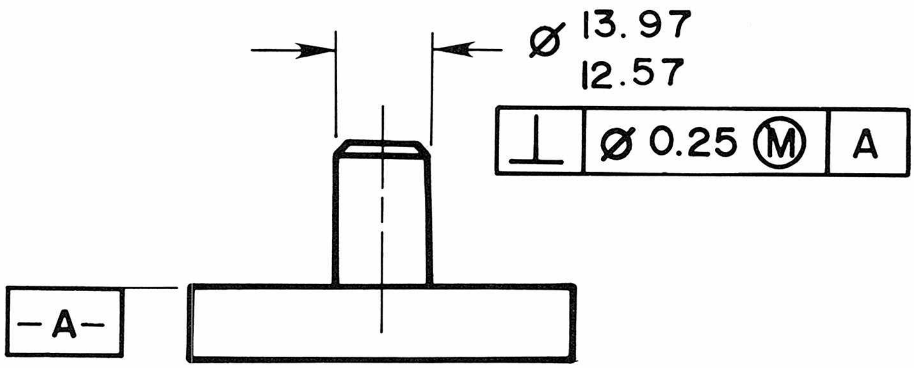
MMC Hole	=	14.22
− Geometric Tolerance	=	0.25
Virtual Condition	=	13.97

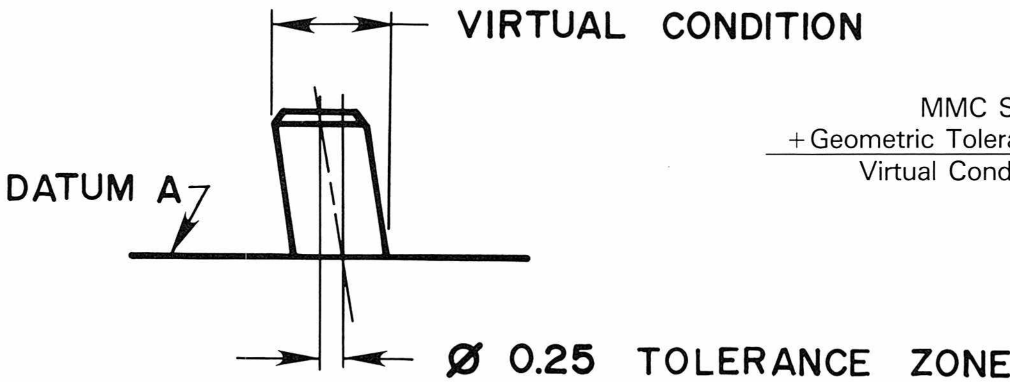
Example 6-18



MMC Hole	=	14.22
− Geometric Tolerance	=	0.25
Virtual Condition	=	13.97

Example 6-19





MMC Shaft	=	13.97
+Geometric Tolerance	=	0.25
Virtual Condition	=	14.22

Example 6-20

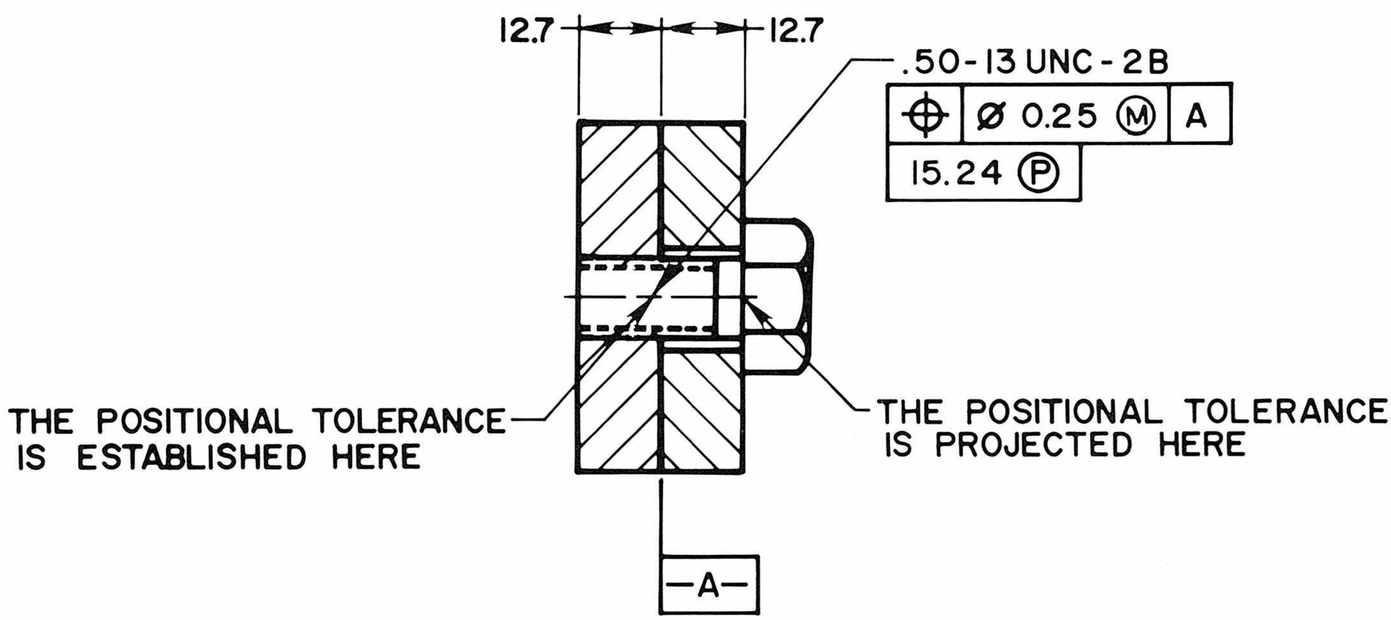
Notice that the virtual condition represents extreme conditions at maximum material condition. This condition is used to determine clearance between mating parts.

PROJECTED TOLERANCE ZONE

At times it may be necessary to control perpendicularity or position from a distance other than from the actual thickness of the part.

A typical situation may be to hold the position of the head of a bolt perpendicular through several thicknesses of material, Example 6-21.

Projected tolerance zone is usually specified for the threaded hole of a fixed fastener application or the press fit hole of a pin application.

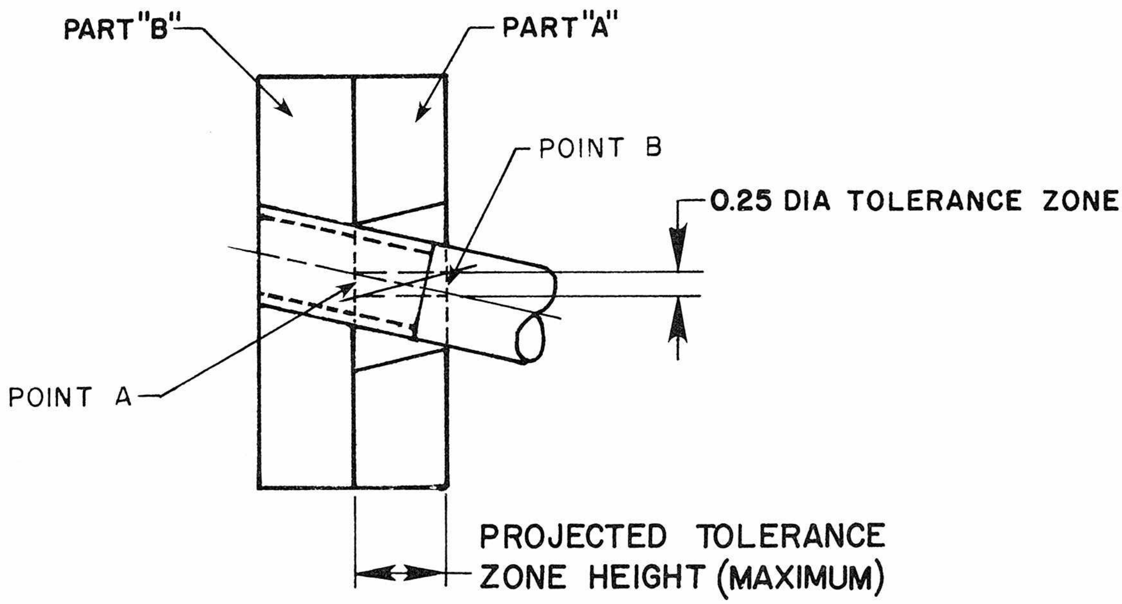


Example 6-21

Example 6-21 shows how a projected tolerance zone should be applied to a feature control symbol.

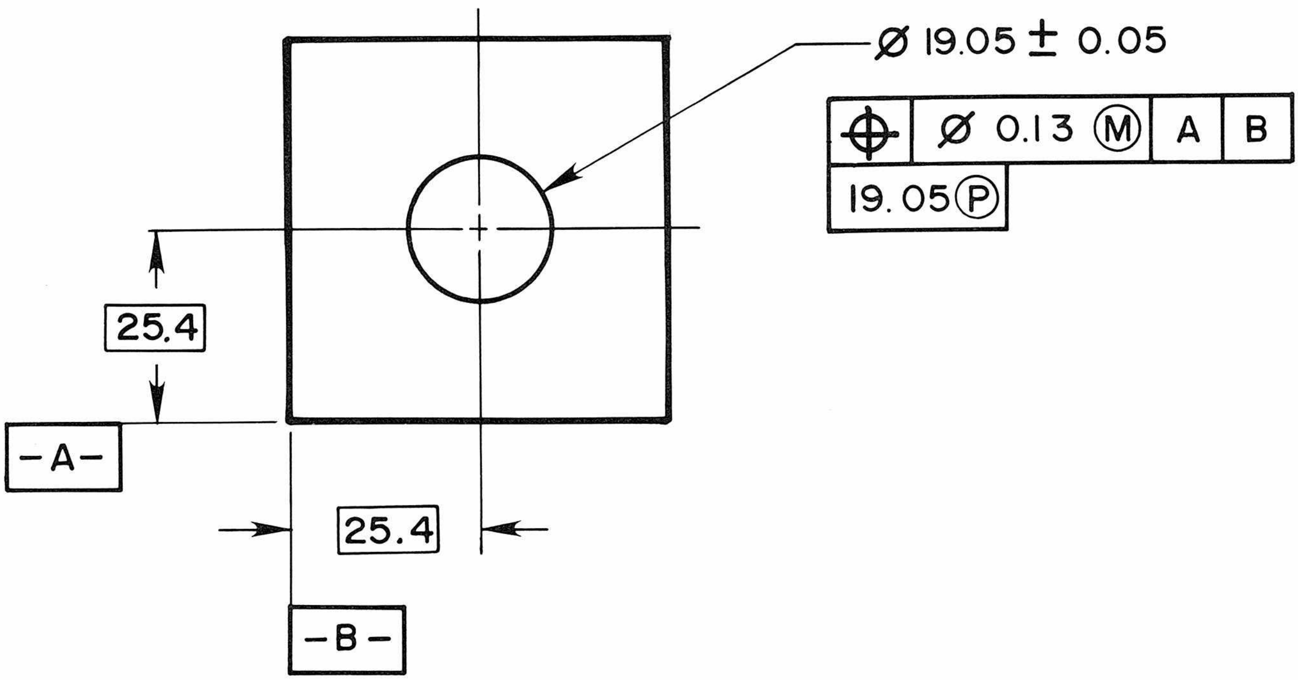
When a projected tolerance zone is not used it is assumed that the geometric or positional tolerance is only applied through the actual thickness of the part.

The drawing in Example 6-22 shows the projected tolerance zone at Example 6-21.



Example 6-22

The location of the threaded hole in Part "B" must begin from Point A in Part "A." Manufacturing begins here and the center may not be out of the projected tolerance zone between Point A and B shown in Part "A." You can see a potential locational area of Part "B."



Example 6-23

The projected tolerance is usually shown with the parts separated and dimensioned as in Example 6-23.

GEOMETRIC DIMENSIONING AND TOLERANCING

Post Test 6 Name:______________________________

1. Provide four short statements that will help characterize position:

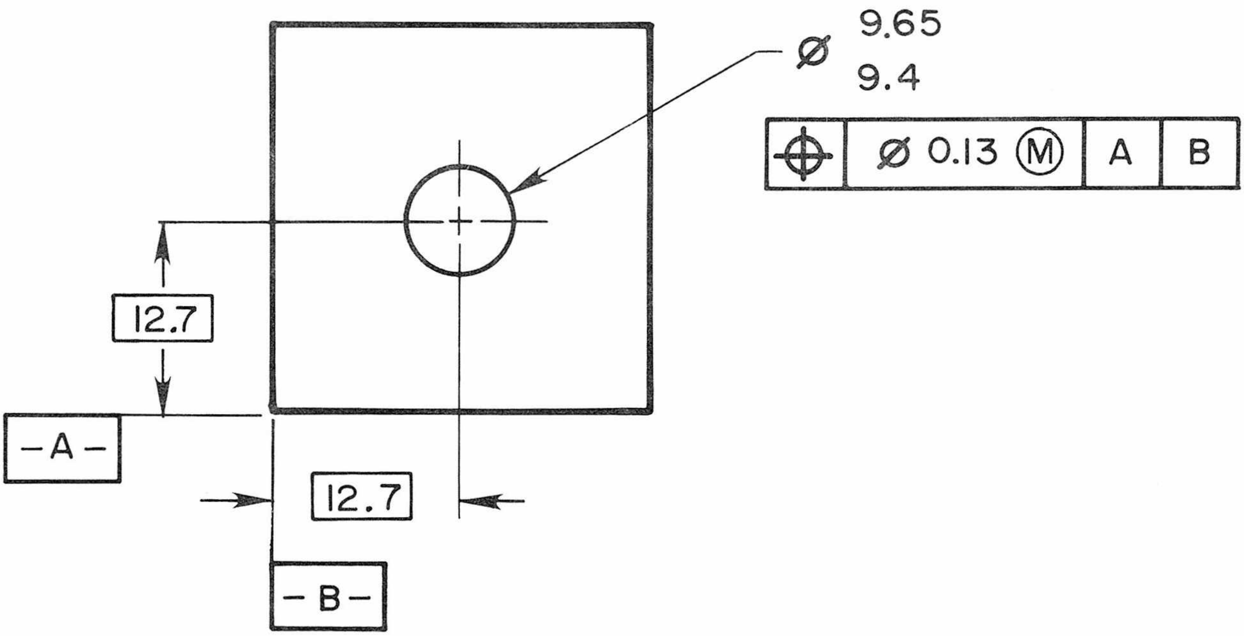
(A)

(B)

(C)

(D)

2. Complete the chart for the object below by indicating how the positional tolerance zone changes as the produced size changes:



Produced Size	Positional Tolerance
9.4	
9.45	
9.50	
9.53	
9.55	
9.60	
9.65	

Post Test 6

3. Given the object below:

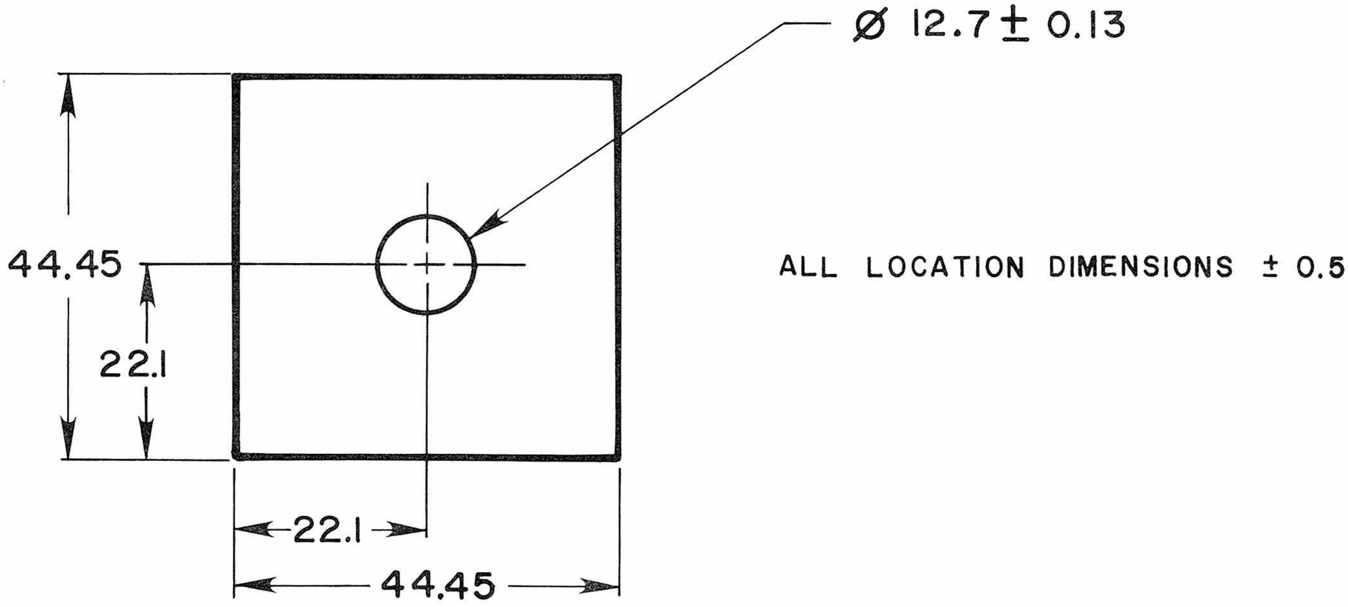
   (A) Calculate the position tolerance zone. (Show your work.)

   (B) Redraw the object on the next page, full scale.

   (C) Properly dimension the hole with position concepts being used.

   (D) Use proper size symbols. Do not sketch.

   (E) Completely convert from conventional location dimensioning to positional location dimensioning.

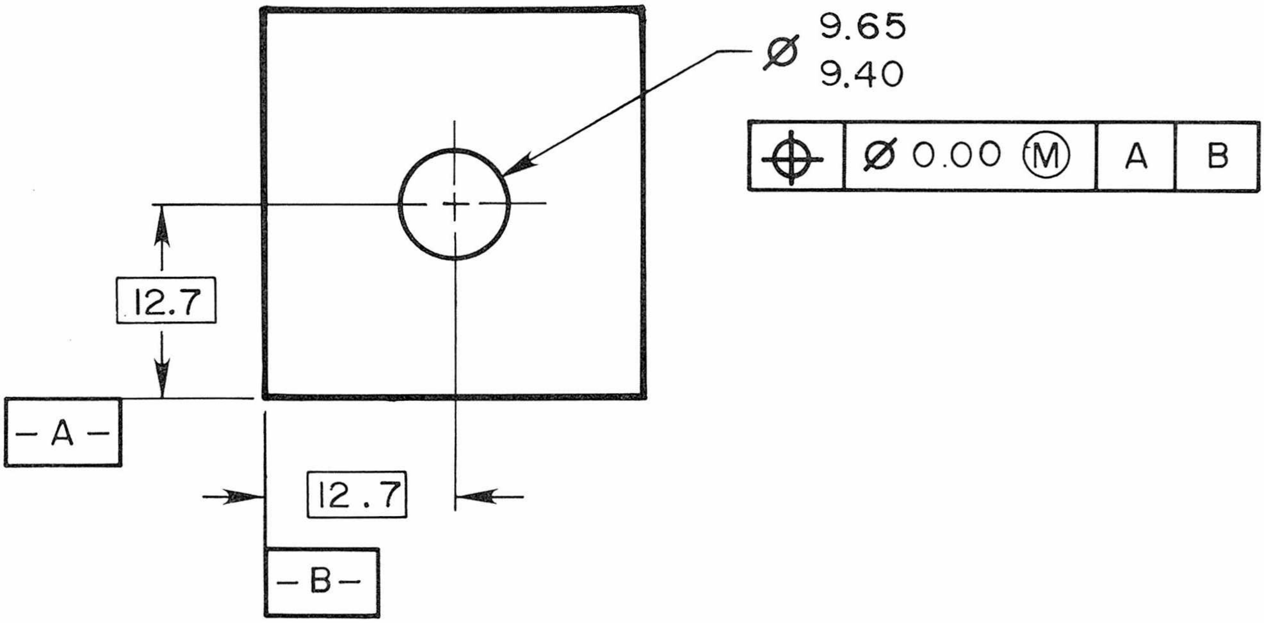


Show your calculations here:

Post Test 6

Post Test 6

4. Complete the chart for the object below by indicating how the positional tolerance zone changes as the produced size changes:



Produced Size	Positional Tolerance
9.40	
9.45	
9.50	
9.53	
9.55	
9.60	
9.65	

Post Test 6

5. Give the formula for:

   (A) Positional tolerance of a floating fastener.

   (B) Positional tolerance of a fixed fastener.

   (C) Virtual condition of a hole.

   (D) Virtual condition of a shaft.

6. What is the virtual condition of the hole through the part in problem No. 2 on page 98? (Show your calculations.)

Post Test 6

7. Define in your own words what is meant by a "projected" tolerance zone.

8. Show an example of a projected tolerance zone by drawing an object in the space below. Completely dimension. Use proper drafting techniques. Make all symbols true size. Do not sketch.

Chapter 7

DIMENSIONING

The current trend in the United States is toward an eventual metric conversion. There are various estimates as to the actual amount of time required for this change. Many companies are using one form of conversion or another. A popular method is to place metric sizes first, followed by inch designations; however, the opposite is also being used. Some companies have gone to a more concise system where all dimensions are in metrics and an inch conversion chart for all dimensions is provided somewhere on the drawing. Often companies have a computer that will create a conversion chart for each drawing. The solution to this problem will be a total conversion to metric dimensioning beginning with design and ending at manufacturing. Until this happens, be prepared to handle a variety of techniques. The following discussion is based upon some current trends for inch/metric dimensioning.

METRICS

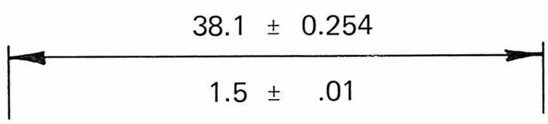
When a drawing has all units in inches or in millimeters it is not necessary to identify the individual units. Currently, with the increased use of metrics, a drawing should contain a general note, similar to the following, that will specify either inches or metrics:

UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN MILLIMETERS (or INCHES as applicable).

The unit of measure accepted by the International System of Units is the millimeter.

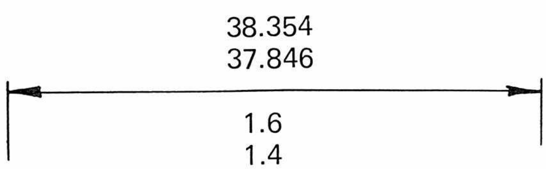
There are several methods of providing for dual dimensioning on a drawing (not covered in ANSI Y14.5). The millimeter can be presented first followed by the inch or the complete opposite method may be used. The actual method used will probably depend upon the standard that your company follows. The best way to reduce confusion is to select a method and stick with it. Example 7-1 shows several methods of dual dimensioning.

(A)

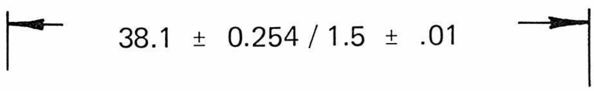


Notice that a zero precedes the decimal point in the millimeter tolerance.

(B)

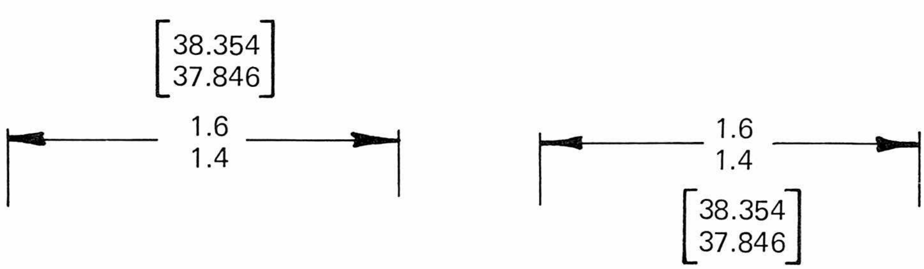


(C)



(D)

(E)



Example 7-1

These methods are only a few of the possibilities, but they demonstrate the common methods.

Dual dimensioning can also be accomplished in a feature control symbol. See Example 7-2.

⊥	0.127/.005	A

or

⊥	.005/0.127	A

or

⊥	0.127 [.005]	A

or

⊥	.005 [0.127]	A

Example 7-2

Dual dimensioning can also be accomplished in a basic dimension symbol as in Example 7-3.

1.5 38.1	or	1.5 [38.1]

Example 7-3

When some inch dimensions are displayed on a metric dimensioned drawing the abbreviation IN. should follow the inch units.

Whichever method is used to dual dimension a drawing, a general note should be indicated to inform the user about the method. See Example 7-4.

$$\frac{\text{MILLIMETER}}{\text{INCH}}\text{ ; MILLIMETER/INCH}$$

or

DIMENSIONS IN [ ] ARE MILLIMETERS

Example 7-4

When you need to convert from inches to millimeters, use this formula:

$$25.4 \times \text{INCH} = \text{MILLIMETER}$$

Convert from millimeters to inches with this formula:

$$\frac{\text{MILLIMETER}}{25.4} = \text{INCH}$$

To achieve the same degree of accuracy when converting from inches to millimeters, the millimeter equivalent should usually be carried out one digit less, to the right of the decimal point, than the inch, Example 7-5.

Inch	Millimeter
.1	2.5
.01	0.3
.001	0.03
.0001	0.003

Example 7-5

NOTE: For additional practice in handling metric conversions on drawings, see Problem 3, page 126.

GEOMETRIC DIMENSIONING AND TOLERANCING

Post Test 7 Name: ______________________

1. Show an example of a general note that will specify that all dimensions are in millimeters.

2. What is the unit of measure accepted by the International System of Units?

3. Convert the following inch dimensions to millimeters.

.016 =	.625 =
.062 =	.750 =
.125 =	1.88 =
.188 =	2.3 =
.250 =	1.8 =
.375 =	3.68 =
.562 =	5.5 =

4. Show five different examples of methods for providing dual dimensioning.

(A)

(B)

(C)

Post Test 7

(D)

(E)

5. Show two examples of dual dimensioning within a feature control frame.

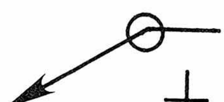
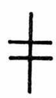
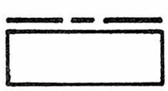
(A)

(B)

6. Show how a method of dual dimensioning can be identified as a general note on a drawing.

SPECIAL SYMBOLS

All symbols are drawn recommended size based on a .125 inch lettering height.

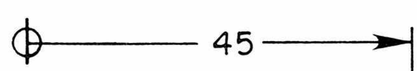
- Ø Diameter
- (75) Reference Dimension
- ╳ Target Point
- ⌾ Dimension Origin
- ⌲ Conical Taper
- ⌳ Slope
- ⌴ Counterbore/Spotface
- ⌵ Countersink
- ↧ Depth/Deep
- □ Square (Shape)
- <u>25</u> Dimension Not To Scale
- 4 X Number of Times/Places
- ⌒86 Arc Length
- R Radius
- SR Spherical Radius
- SØ Spherical Diameter
-  All Around
-  Symmetrical
-  Chain Line

SOME COMMON SYMBOL USES

1. 8X Ø 8.4
   ⌴ Ø 12.7
   ↧ 5.2

2. 6 X Ø $12^{+0.27}_{\ \ 0}$

3. Ø 6.2 THRU
   ⌵ Ø 12 X 82°

4. 

INCHES: FRACTIONS	INCHES: DECIMALS	MILLIMETERS
	.00394	.1
	.00787	.2
	.01181	.3
1/64	.015625	.3969
	.01575	.4
	.01969	.5
	.02362	.6
	.02756	.7
1/32	.03125	.7938
	.0315	.8
	.03543	.9
	.03937	1.00
3/64	.046875	1.1906
1/16	.0625	1.5875
5/64	.078125	1.9844
	.07874	2.00
3/32	.09375	2.3813
7/64	.109375	2.7781
	.11811	3.00
1/8	.125	3.175
9/64	.140625	3.5719
5/32	.15625	3.9688
	.15748	4.00
11/64	.171875	4.3656
3/16	.1875	4.7625
	.19685	5.00
13/64	.203125	5.1594
7/32	.21875	5.5563
15/64	.234375	5.9531
	.23622	6.00
1/4	.2500	6.35
17/64	.265625	6.7469
	.27559	7.00
9/32	.28125	7.1438
19/64	.296875	7.5406
5/16	.3125	7.9375
	.31496	8.00
21/64	.328125	8.3344
11/32	.34375	8.7313
	.35433	9.00
23/64	.359375	9.1281
3/8	.375	9.525
25/64	.390625	9.9219
	.3937	10.00
13/32	.40625	10.3188
27/64	.421875	10.7156
	.43307	11.00
7/16	.4375	11.1125
29/64	.453125	11.5094
15/32	.46875	11.9063
	.47244	12.00
31/64	.484375	12.3031
1/2	.5000	12.70
	.51181	13.00
33/64	.515625	13.0969
17/32	.53125	13.4938
35/64	.546875	13.8907
	.55118	14.00
9/16	.5625	14.2875
37/64	.578125	14.6844
	.59055	15.00
19/32	.59375	15.0813
39/64	.609375	15.4782
5/8	.625	15.875
	.62992	16.00
41/64	.640625	16.2719
21/32	.65625	16.6688
	.66929	17.00
43/64	.671875	17.0657
11/16	.6875	17.4625
45/64	.703125	17.8594
	.70866	18.00
23/32	.71875	18.2563
47/64	.734375	18.6532
	.74803	19.00
3/4	.7500	19.05
49/64	.765625	19.4469
25/32	.78125	19.8438
	.7874	20.00
51/64	.796875	20.2407
13/16	.8125	20.6375
	.82677	21.00
53/64	.828125	21.0344
27/32	.84375	21.4313
55/64	.859375	21.8282
	.86614	22.00
7/8	.875	22.225
57/64	.890625	22.6219
	.90551	23.00
29/32	.90625	23.0188
59/64	.921875	23.4157
15/16	.9375	23.8125
	.94488	24.00
61/64	.953125	24.2094
31/32	.96875	24.6063
	.98425	25.00
63/64	.984375	25.0032
1	1.0000	25.4001

Metric – inch equivalents.

CONVERSION TABLE
ENGLISH TO METRIC

WHEN YOU KNOW	MULTIPLY BY: * = Exact — VERY ACCURATE	MULTIPLY BY: * = Exact — APPROXIMATE	TO FIND
LENGTH			
inches	* 25.4		millimeters
inches	* 2.54		centimeters
feet	* 0.3048		meters
feet	* 30.48		centimeters
yards	* 0.9144	0.9	meters
miles	* 1.609344	1.6	kilometers
WEIGHT			
grains	15.43236	15.4	grams
ounces	* 28.349523125	28.0	grams
ounces	* 0.028349523125	.028	kilograms
pounds	* 0.45359237	0.45	kilograms
short ton	* 0.90718474	0.9	tonnes
VOLUME			
teaspoons		5.0	milliliters
tablespoons		15.0	milliliters
fluid ounces	29.57353	30.0	milliliters
cups		0.24	liters
pints	* 0.473176473	0.47	liters
quarts	* 0.946352946	0.95	liters
gallons	* 3.785411784	3.8	liters
cubic inches	* 0.016387064	0.02	liters
cubic feet	* 0.028316846592	0.03	cubic meters
cubic yards	* 0.764554857984	0.76	cubic meters
AREA			
square inches	* 6.4516	6.5	square centimeters
square feet	* 0.09290304	0.09	square meters
square yards	* 0.83612736	0.8	square meters
square miles		2.6	square kilometers
acres	* 0.40468564224	0.4	hectares
TEMPERATURE			
Fahrenheit	* 5/9 (after subtracting 32)		Celsius

CONVERSION TABLE
METRIC TO ENGLISH

WHEN YOU KNOW	MULTIPLY BY: * = Exact — VERY ACCURATE	MULTIPLY BY: * = Exact — APPROXIMATE	TO FIND
LENGTH			
millimeters	0.0393701	0.04	inches
centimeters	0.3937008	0.4	inches
meters	3.280840	3.3	feet
meters	1.093613	1.1	yards
kilometers	0.621371	0.6	miles
WEIGHT			
grains	0.00228571	0.0023	ounces
grams	0.03527396	0.035	ounces
kilograms	2.204623	2.2	pounds
tonnes	1.1023113	1.1	short tons
VOLUME			
milliliters		0.2	teaspoons
milliliters	0.06667	0.067	tablespoons
milliliters	0.03381402	0.03	fluid ounces
liters	61.02374	61.024	cubic inches
liters	2.113376	2.1	pints
liters	1.056688	1.06	quarts
liters	0.26417205	0.26	gallons
liters	0.03531467	0.035	cubic feet
cubic meters	61023.74	61023.7	cubic inches
cubic meters	35.31467	35.0	cubic feet
cubic meters	1.3079506	1.3	cubic yards
cubic meters	264.17205	264.0	gallons
AREA			
square centimeters	0.1550003	0.16	square inches
square centimeters	0.00107639	0.001	square feet
square meters	10.76391	10.8	square feet
square meters	1.195990	1.2	square yards
square kilometers		0.4	square miles
hectares	2.471054	2.5	acres
TEMPERATURE			
Celsius	*9/5 (then add 32)		Fahrenheit

MEASURING SYSTEMS

ENGLISH	METRIC
LENGTH	
12 inches = 1 foot	1 kilometer = 1000 meters
36 inches = 1 yard	1 hectometer = 100 meters
3 feet = 1 yard	1 decameter = 10 meters
5,280 feet = 1 mile	1 meter = 1 meter
16.5 feet = 1 rod	1 decimeter = 0.1 meter
320 rods = 1 mile	1 centimeter = 0.01 meter
6 feet = 1 fathom	1 millimeter = 0.001 meter
WEIGHT	
27.34 grains = 1 dram	1 tonne = 1,000,000 grams
438 grains = 1 ounce	1 kilogram = 1000 grams
16 drams = 1 ounce	1 hectogram = 100 grams
16 ounces = 1 pound	1 dekagram = 10 grams
2000 pounds = 1 short ton	1 gram = 1 gram
2240 pounds = 1 long ton	1 decigram = 0.1 gram
25 pounds = 1 quarter	1 centigram = 0.01 gram
4 quarters = 1 cwt	1 milligram = 0.001 gram
VOLUME	
8 ounces = 1 cup	1 hectoliter = 100 liters
16 ounces = 1 pint	1 decaliter = 10 liters
32 ounces = 1 quart	1 liter = 1 liter
2 cups = 1 pint	1 deciliter = 0.1 liter
2 pints = 1 quart	1 centiliter = 0.01 liter
4 quarts = 1 gallon	1 milliliter = 0.001 liter
8 pints = 1 gallon	1000 milliliter = 1 liter
AREA	
144 sq. inches = 1 sq. foot	100 sq. millimeters = 1 sq. centimeter
9 sq. feet = 1 sq. yard	100 sq. centimeters = 1 sq. decimeter
43,560 sq. ft. = 160 sq. rods	100 sq. decimeters = 1 sq. meter
160 sq. rods = 1 acre	10,000 sq. meters = 1 hectare
640 acres = 1 sq. mile	

TEMPERATURE

FAHRENHEIT		CELSIUS (centigrade)
32 degrees F	Water freezes	0 degrees C
68 degrees F	Reasonable room temperature	20 degrees C
98.6 degrees F	Normal body temperature	37 degrees C
173 degrees F	Alcohol boils	78.34 degrees C
212 degrees F	Water boils	100 degrees C

BASIC GEOMETRIC DIMENSIONING AND TOLERANCING

FINAL EXAM – Part 1 Name: ____________________________

Below (left) is a list of short descriptions. A list of words or symbols is at the right. Place the letter of the word or symbol that matches the description in the blank provided. Each letter may be used more than once.

_____ 1. The tolerance zone of a geometric characteristic that must have a material condition symbol MMC, RFS, or LMC.

_____ 2. Datum identification symbol.

_____ 3. All datum planes on a part intersecting at right angles are 90° BASIC by interpretation.

_____ 4. A form control within a tolerance zone between two concentric cylinders.

_____ 5. A geometric characteristic that has no deviation of specified tolerance from a straight line.

_____ 6. A geometric characteristic with a tolerance zone between two parallel planes and perpendicular to a datum.

_____ 7. A geometric characteristic with a tolerance zone between two parallel planes and parallel to a datum.

_____ 8. Condition of a part wherein it contains maximum amount of MATERIAL.

_____ 9. Implies RFS. (Select an answer other than Ⓢ.)

_____ 10. RFS.

_____ 11. This geometric characteristic requires a basic angularity dimension.

_____ 12. This geometric characteristic is used to identify a tolerance of location.

_____ 13. A single element form control that specifies a tolerance zone between two concentric circles.

_____ 14. A feature is permitted only the stated position or form tolerance; no change as size varies.

_____ 15. Dimensions orginating from a datum, control location of _________.

(A) ⊥

(B) Datum target symbol

(C) Ⓢ

(D) //

(E) .010

(F) Related features

(G) Sizes

(H) ○

(I) Ⓜ

(J) ⌖

(K) .176

(L) $\boxed{\text{-A-}}$

(M) ⌭

(N) —

(O) Form

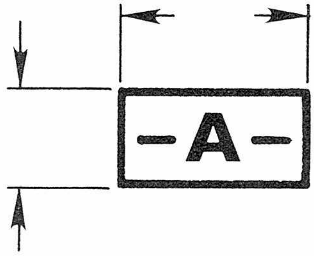
(P) Three plane concept

(Q) ∠

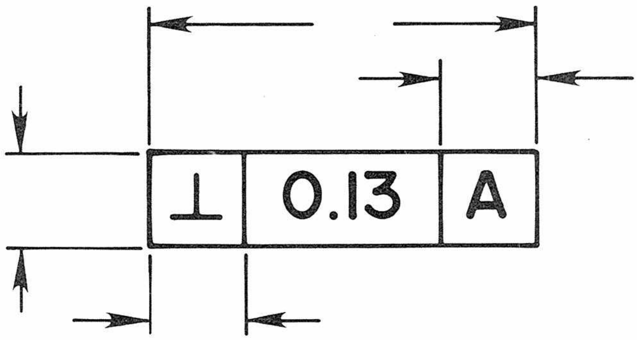
FINAL EXAM – Part 2, Symbols

Answer the following questions:

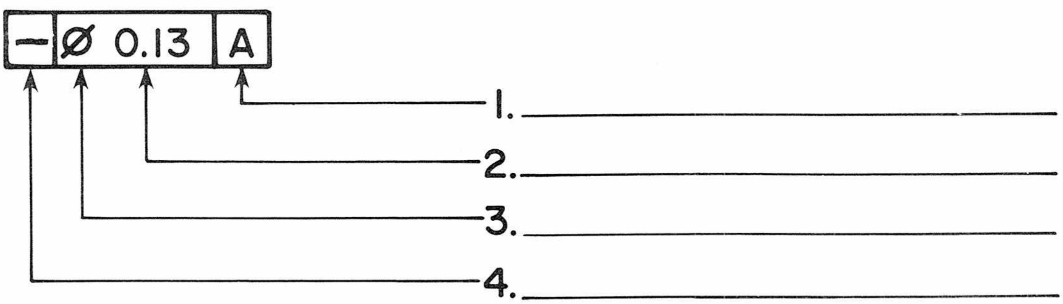
1. The symbol below is called ________________________________ and it would be drawn the size shown below: (Fill in the dimensions.)



2. The symbol below is called ________________________________ and it would be drawn the size below. (Fill in the dimensions.)



3. Identify the items in the symbol below and write your answers in the four blanks on the right.



FINAL EXAM

4. List the symbols for the geometric characteristics and indicate the symbol name.

5. What does this symbol represent?

____

6. List three items that may be identified as datums.

____

____

____

7. What is the symbol for each of the items listed below?

Maximum Material Condition ______________________________

Regardless of Feature Size ______________________________

Projected Tolerance Zone ______________________________

Least Material Condition ______________________________

FINAL EXAM — Part 3

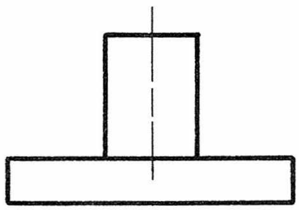
1. What is the MMC of a hole with a diameter of 15.875 ± 0.127?

   ____

2. What is the MMC of a shaft with a diameter of 15.875 ± 0.127?

   ____

3. Properly dimension the following object so that the 12.7 ± 0.127 diameter shaft will be perpendicular to the top surface of the base by a MMC tolerance of 0.254.



4. If the object in No. 3 were RFS rather than MMC, would the RFS symbol be necessary? Why?

   ____

   ____

5. Fill in the blanks of the chart below for the axis control of following specification.

   $\varnothing 9.525\,^{+0.00}_{-0.127}$

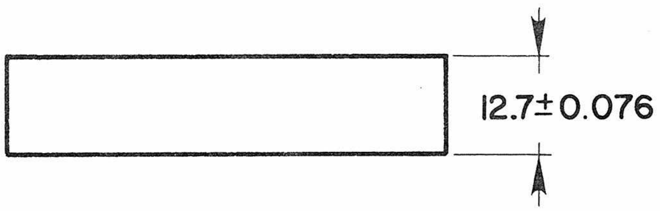
Actual Size	— | Ø 0.03 | A	— | Ø0.03 Ⓜ | A
MMC 9.525	________	________
9.449	________	________
9.436	________	________
9.398	________	________

6. Fill in the blanks of the chart below for the following specification:
   Ø15.860 ± 0.013 shaft.

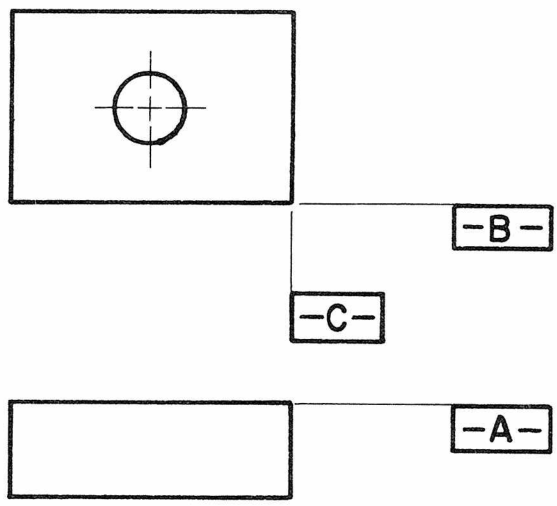
Actual Size	⊥ | Ø0.01 | A	⊥ | Ø0.01 Ⓜ | A
MMC 15.873	________	________
15.868	________	________
15.864	________	________
15.855	________	________
15.850	________	________
15.847	________	________

FINAL EXAM

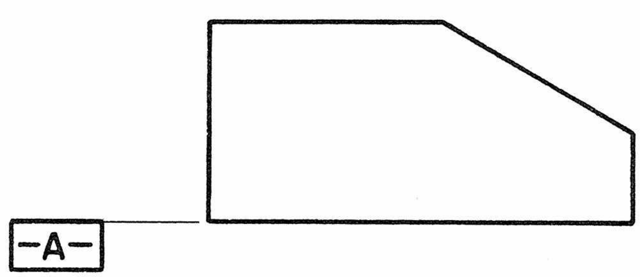
7. Place a 0.025 flatness callout on the bottom surface of the object below.



8. On the drawing below, give the basic dimension to locate the hole from datums -C- and -B-. A diameter callout of 6.35 ± 0.05 and perpendicular by 0.025 to datum -A- is required. Make datum -B- perpendicular to datum -C- by 0.05 and datum -A- will be flat by 0.05.

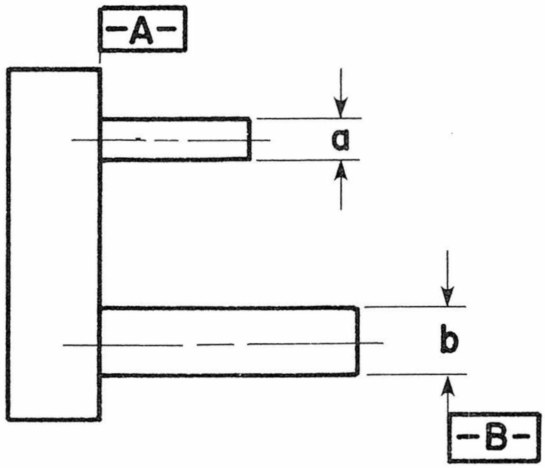


9. Show that the angle below will be within 0.127 of datum -A-. The slanted surface is 30° from datum -A-.

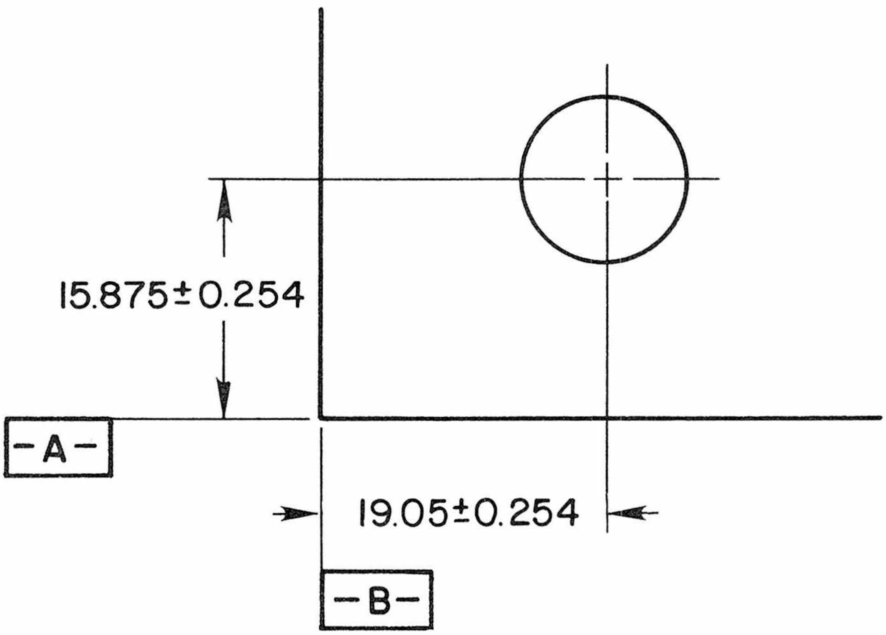


FINAL EXAM

10. On the object below provide these callouts; diameter "a" will be parallel to datum -B- by 0.05 and diameter "b" will be perpendicular to -A- by 0.025.



11. Find the maximum positional displacement of the 12.7 ± 0.025 diameter hole and give the proper positional and locational callouts to -A- and -B-. (Show your calculations.)

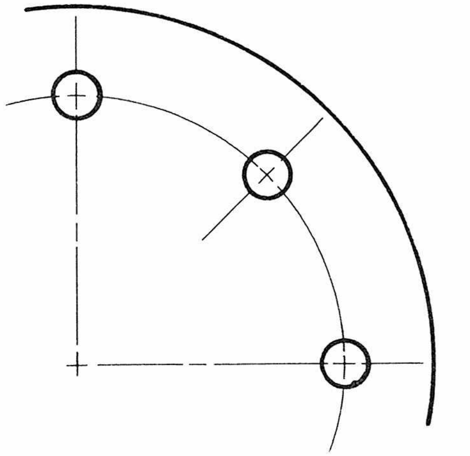


Will the ± tolerance on the location dimensions above remain? ____________ Explain: ____________

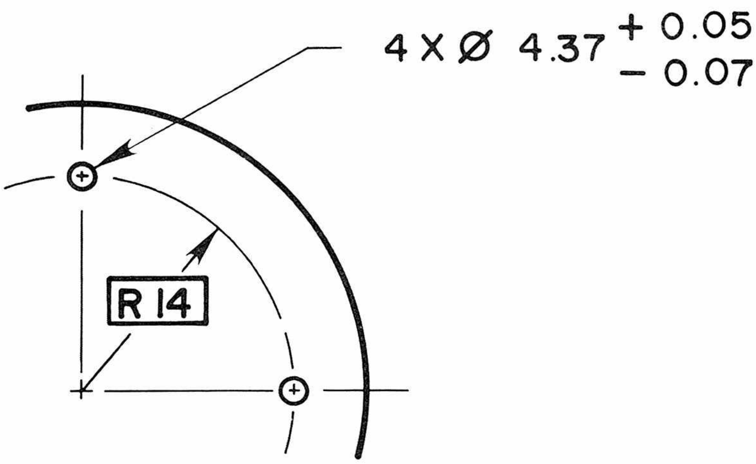
____

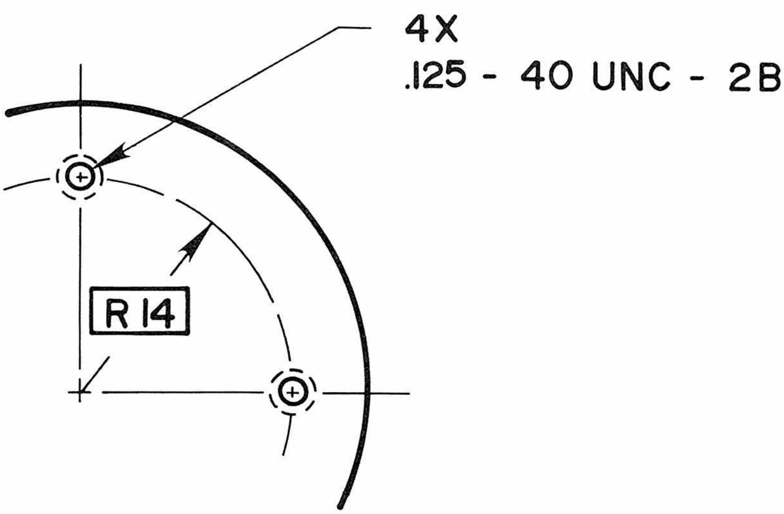
FINAL EXAM

12. Calculate the positional tolerance of the 6.477 ± 0.076 holes below so that you can use .250 (6.35 mm) diameter machine screws to bolt to the mating parts. This is a floating fastener. Dimension directly on the object below. (Show your calculations.)



13. Calculate the positional tolerance necessary to insure mating of the two parts below. (Show your calculations.) Then, provide the proper feature control symbol to each part. Assume both parts are 12.7 thick project tolerance zone at threaded features.

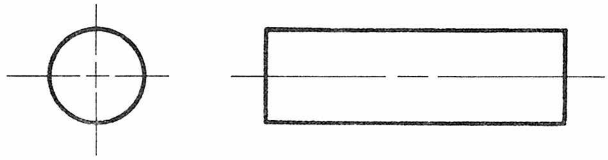




The object in Problem No. 13 is a ______________ fastener.

FINAL EXAM

14. Provide a 0.254 radius tolerance zone for roundness on the object below.



15. Provide a general note that would specify a metric dimensioned drawing.

FINAL EXAM – Part 4

1. If datum plane $\boxed{\text{-A-}}$ is primary, datum plane $\boxed{\text{-B-}}$ is secondary, and datum plane $\boxed{\text{-C-}}$ is tertiary, then:

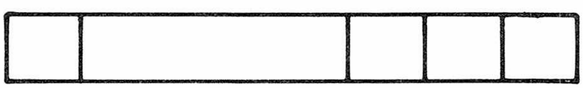
   (A) How many points, minimum, are required to establish each of the datums?

   $\boxed{\text{-A-}}$ ______________________________

   $\boxed{\text{-B-}}$ ______________________________

   $\boxed{\text{-C-}}$ ______________________________

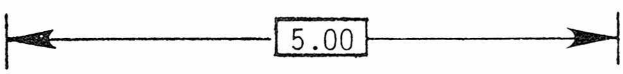
   (B) Complete the symbol below by filling in all elements in correct order.



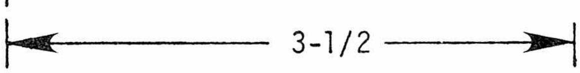
2. Does a basic dimension have a tolerance?

   Yes   No

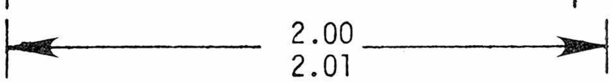
3. Which of the dimensions shown below are basic dimensions?

(A) 

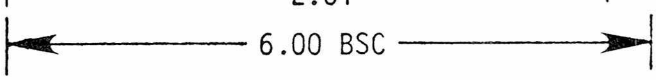
Yes [ ] No [ ]

(B) 

Yes [ ] No [ ]

(C) 

Yes [ ] No [ ]

(D) 

Yes [ ] No [ ]

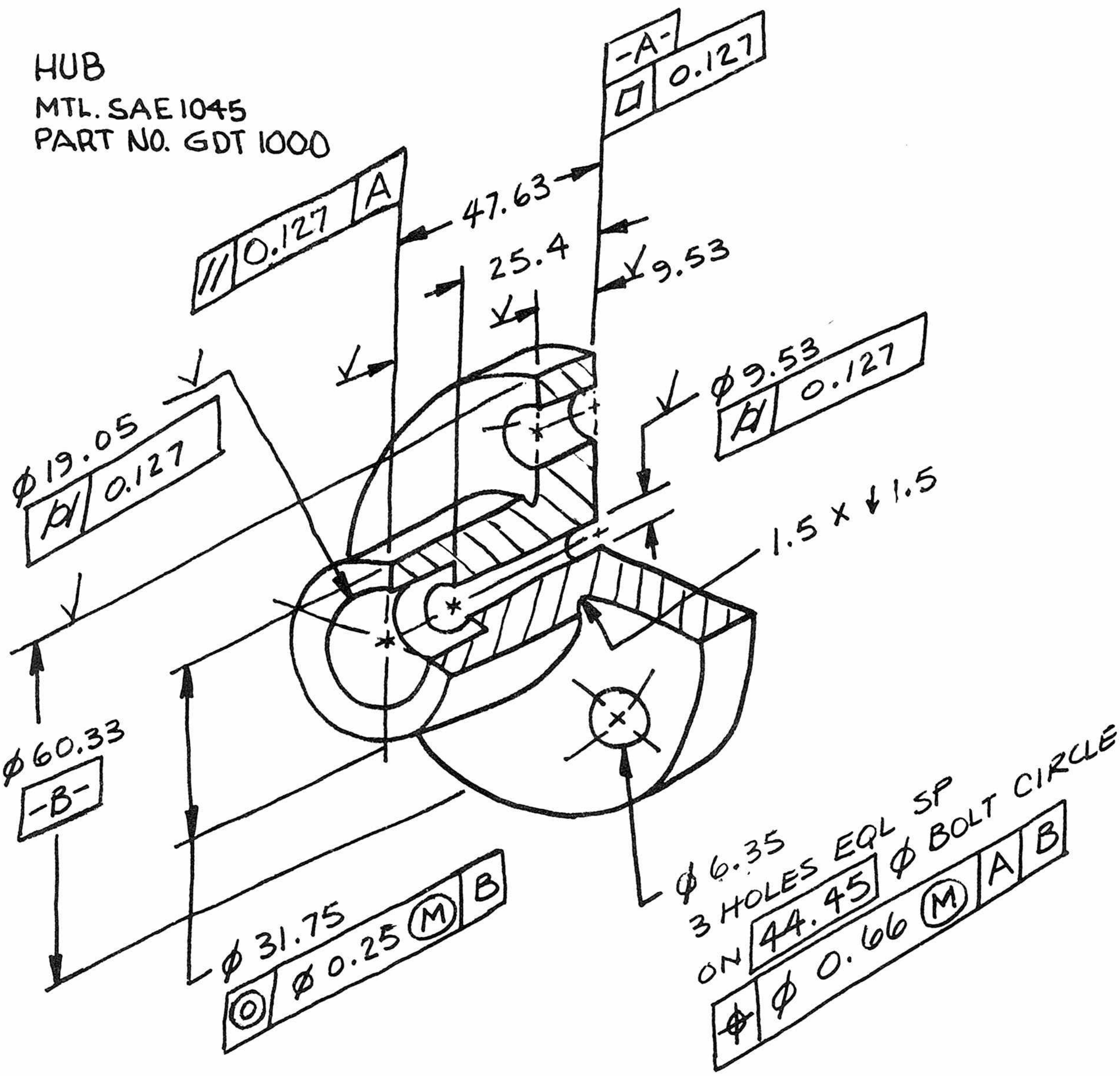
DRAWING PROBLEMS

DRAWING PROBLEM 1

This drawing should be completed after Chapter 3. First, read all instructions.

Given an engineer's sketch, draw the object on "C" size vellum (17 x 22 or 18 x 24) using pencil as a medium. Draw all necessary views and completely dimension as shown on the sketch (not necessarily placement). Use dimensions as given. Use unidirectional dimensioning. All dimensions are in millimeters.

Your grade will be based on:
1. Layout.
2. View placement.
3. Accuracy.
4. Dimensioning practice and proper use of geometric dimensioning and tolerancing.
5. Line quality.
6. Lettering.
7. Neatness.



DRAWING PROBLEM 2

This drawing should be completed after Chapter 5. Read all instructions before you begin.

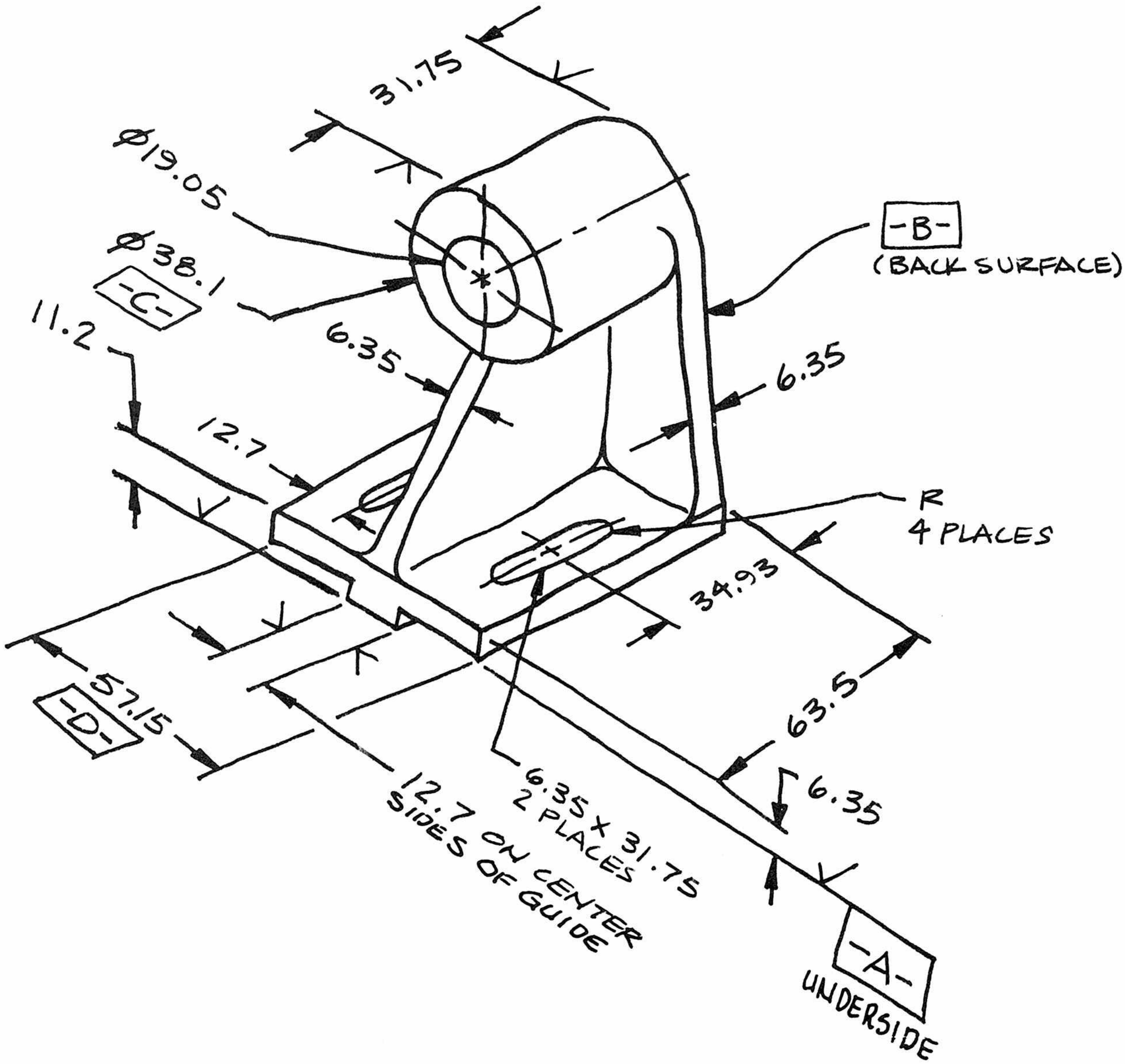
Given an Engineering Sketch, draw the object on ''C'' size vellum (17 x 22 or 18 x 24) using pencil as a medium. Draw all necessary views and completely dimension. Dimensions are given in millimeters.

The following geometric dimensions should be applied to the object.

1. Label all datums.
2. Use dimensions shown on the sketch (not necessarily placement).
3. Use unidirectional dimensioning.
4. The center line of the 19.05 diameter is 65.07 from datum -A-.
5. Datum -B- perpendicular to datum -A- by 0.05.
6. Sides of guide (12.7) shall be perpendicular to datum -A- by 0.05 and perpendicular to datum -B- by 0.127.
7. Bottom surface of guide (12.7) shall be parallel to datum -A- by 0.127 and perpendicular to datum -B- by 0.127.
8. The web will form an angularity tolerance to datum -A- by 2.54, RFS.
9. 19.05 diameter will be perpendicular to datum -B- by 0.71, MMC.
10. Two (2) slots will be perpendicular to datum -A- by 0.127, MMC.

Your grade will be based on:

1. Layout.
2. View placement.
3. Accuracy.
4. Dimensioning practice and proper use of geometric dimensioning and tolerancing.
5. Line quality.
6. Lettering.
7. Neatness.



IMPORTANT INFORMATION

1. Break all sharp corners.
2. Fillets and Rounds R 3
3. MTL: SAE 1030 Steel
4. Unless otherwise specified, dimensions are millimeters.
5. Part No. GDT 2000

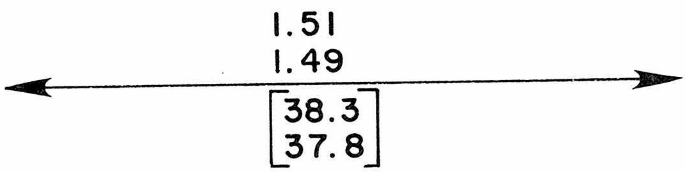
PEDESTAL

DRAWING PROBLEM 3

This drawing should be completed after Chapter 7. Read all instructions before you begin.

The following sketch is a latch bracket that will be used on the hatch for the spacecraft going to Mars. Make an appropriate multiview drawing of the latch bracket on "C" size vellum (17 x 22 or 18 x 24). Dimensions are in inches.

1. Label all datums.
2. Use dimensions as shown on the sketch (not necessarily placement).
3. Provide angularity tolerance of .005 to datum -C- for 70° angle.
4. Hold surface profile of .005 between points X and Y, two places at the 1.25 Rs.
5. C'Bore position to datums A, B, and C, by .003 MMC.
6. Surfaces 3 and 4 parallel with surfaces 1 and 2 by .004.
7. .187 DIA position with datums A, B, C by .001 MMC and projected 2.00 from -A-.
8. .86/.89 DIA perpendicular to datum -E- by .005 projected 1.68.
9. Datum -B- perpendicular to datum -A- by .002.
10. Datum -C- perpendicular to datum -A- and -B- by .002.
11. Material titanium.
12. Use dual dimensioning, with inches followed by millimeters in brackets. See Example 3000-1.
13. Part number GDT 3000.

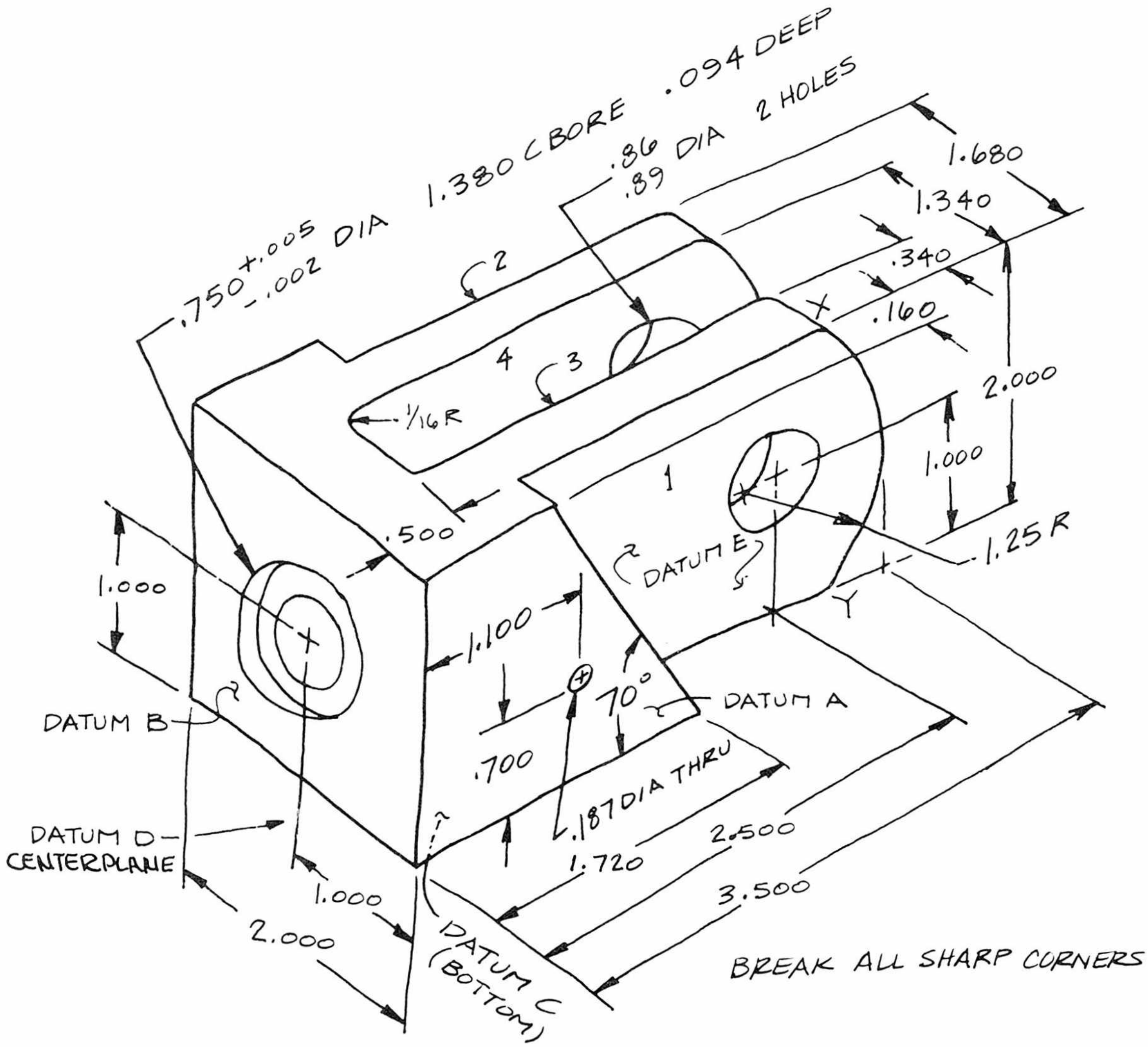


Example 3000-1

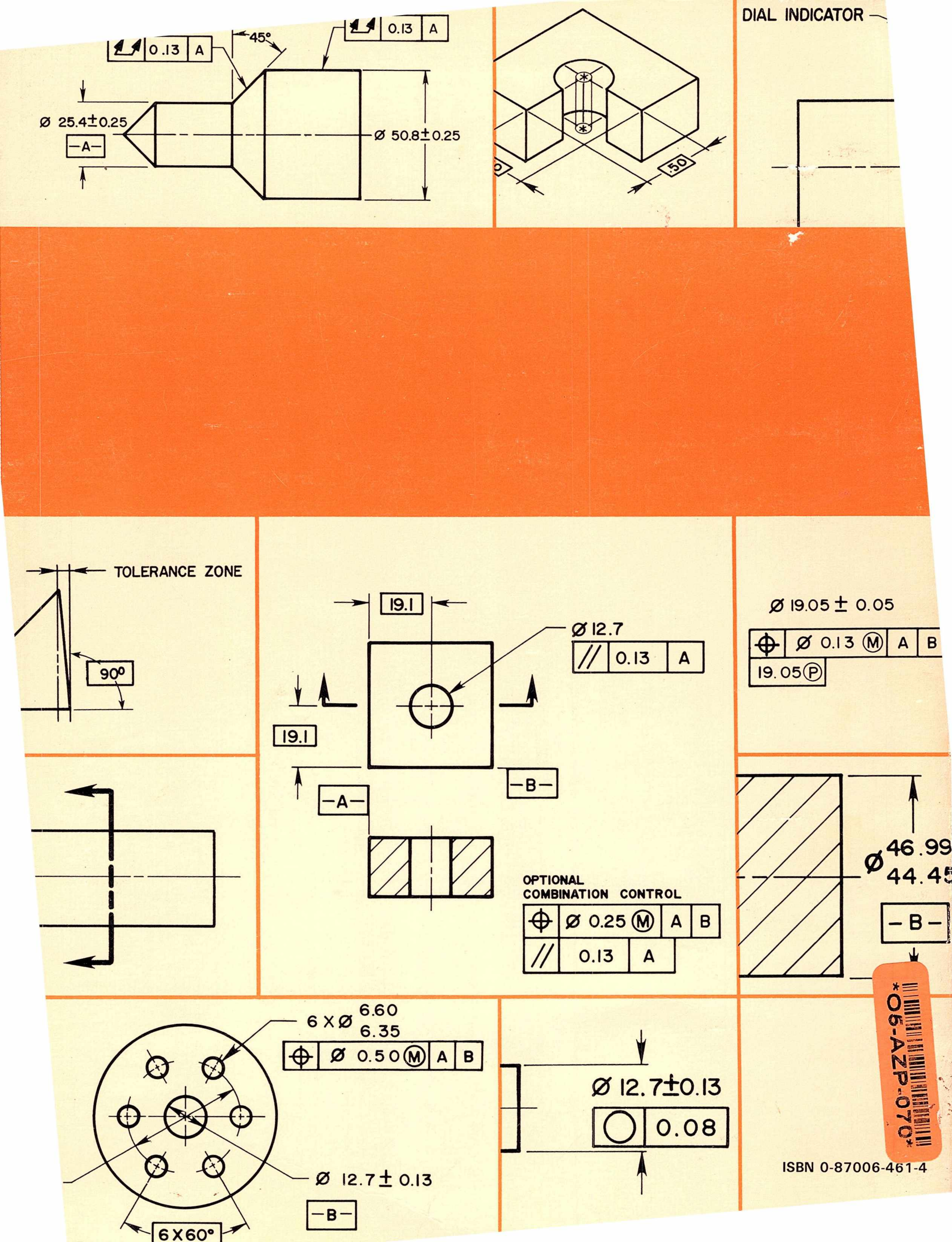
14. Provide the appropriate metric general note for dual dimensioning.

Your grade will be based on:

1. Layout.
2. View placement.
3. Accuracy.
4. Dimensioning practice and proper use of geometric dimensioning and tolerancing.
5. Line quality.
6. Lettering.
7. Neatness.


.750 +.005 -.002 DIA 1.380 CBORE .094 DEEP
.86 .89 DIA 2 HOLES
1.680
1.340
.340
.160
X
2
4
3
1/16R
2.000
1.000
1
1.25R
.500
DATUM E
Y
1.000
1.100
70°
DATUM A
DATUM B
.700
.187 DIA THRU
DATUM D CENTERPLANE
1.720
2.500
3.500
1.000
2.000
DATUM C (BOTTOM)
BREAK ALL SHARP CORNERS

7



ISBN 0-87006-461-4